Haas Factory Outlet

Haas Mill Series Training Manual

Advanced Programming Techniques

Productivity Inc

For more information on Additional Training Opportunities or our Classroom Schedule,
Contact the Productivity Inc Applications Department in Minneapolis:
☏ 763.476.8600

Visit us on the Web: www.productivity.com

Click on the Training Registration Button

✉ trainingmn@productivity.com

Advanced Programming Techniques – Table of Contents

Advanced Haas Program Techniques

Haas Programmer Optimizer

The Haas Program Optimizer allows feed and speed overrides, coolant P changes, notes to be saved after a program has been run for the first time. First the program is run in memory and any changes to speed or feed are made thru the override keys. If the coolant position is not correct usually the machine is put on Feed Hold and the P coolant position is adjusted using the CLNT UP or CLNT DOWN keys. If a coolant needs to be turned on or off the machine is put on Feed Hold and the COOLNT key on the MDI mode line pressed. A note can be made such as ADD PASS and the Enter key depressed. If a M01 needs to be changed to an M00, this can also be noted.

To get into the Program Optimizer press the **F4** key at the end of the program. The following screen will come up in the Edit Mode. Toggling the EDIT key will make one or the other the active screen in white. Note on blocks N5 thru N7 have asterisks on them. This indicates some change was made while the program was run or a note was added.

Pressing the EDIT key will make the right side of the screen active. Then highlight in yellow the F 110% and press the ENTER key. The following pop will appear. Using the up and down cursor key gives different options. Text below the line elaborates on the different selections. The Alter Feed on current line was highlighted in yellow and selected. Pressing the Enter key will alter the feed on the current line.

```
FEED AND SPEED OVERRIDE            CANCEL - Exit
  Alter previous FEED
  Insert previous FEED on current line
  Alter FEED on current line
  Alter all FEEDs up to previous Tool
  Alter all FEEDs down to next Tool
  Press CANCEL to exit.

Alters the FEED on the current line with the
current override percentage from the highlighted
location in the Optimizer.
```
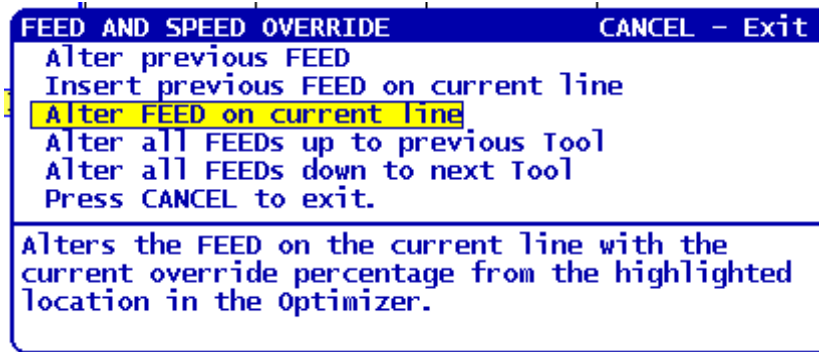
The following pop up appears giving what the override feed will be.   To change the feed press the ALTER key. The feed on Line 5 is changed to 55 and the old feed rate F50 is put in parenthesis in the program.

```
FEED AND SPEED OVERRIDE

The Current Feed is 5.

At 110% override the new feed is 5.5

To insert on current line press ALTER.

To ignore press CANCEL
```
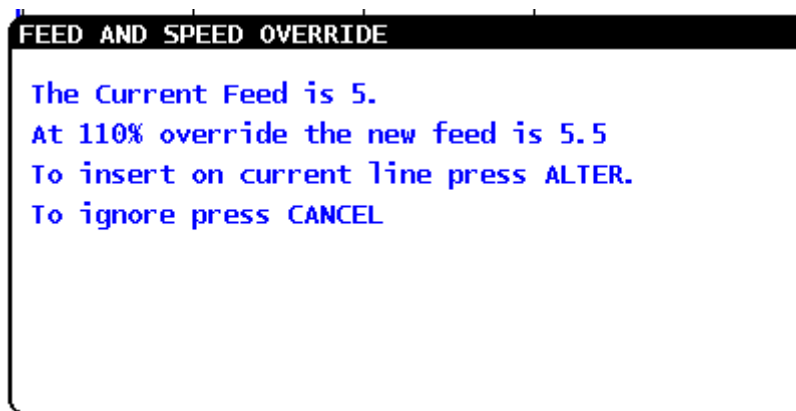
On line N6 the speed override is highlighted and the Enter key is pressed. A similar pop up appears for the Speed Override.   See below:

```
FEED AND SPEED OVERRIDE            CANCEL - Exit
  Alter previous SPEED
  Insert previous SPEED on current line
  Alter SPEED on current line
  Alter SPEEDs up to previous Tool
  Alter all SPEEDs down to next Tool
  Press CANCEL to exit.

Alters the previous SPEED with the current
override percentage from the highlighted
location in the Optimizer.
```

Alter previous SPEED was selected and confirmation pop appears below. Alter key is pressed.

```
FEED AND SPEED OVERRIDE

  The Current Speed is 4000
  The Override Speed is 3600


  To accept press ALTER.
  To ignore press CANCEL
```
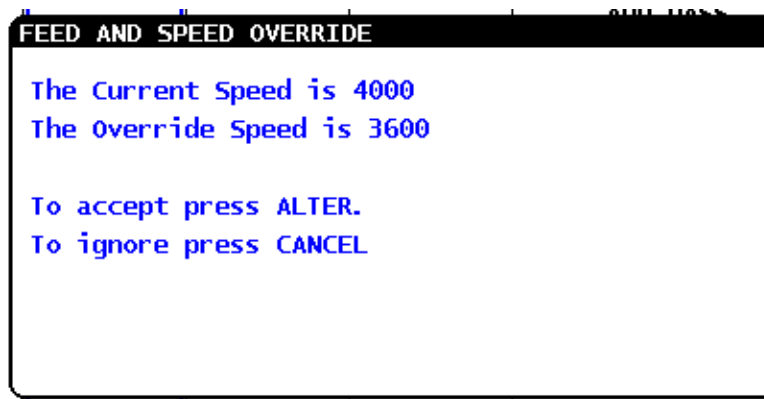
The following gives the edits which have been made to the original code. The new Speed and Feed are changed and the old speed and feed rates are put in parenthesis.

```
N3 G00 G90 G54 X3.25 Y0 S3600 (S4000) M03 ;
N4 G43 H01 Z0.1 ;
*N5 G01 Z0 F55. (F50.) ;
*N6 X-3.25 ;
*N7 G00 Z0.1 M09 ;
```

Cursor to the M08 on block N41.

```
                                    EDIT: EDIT
ACTIVE PROGRAM – 000010              Feed %   Spindle %  Coolant   Cool Pos      Notes
 000010 (CLASS) ;
 N1 T1 M06 (2 INCH FACE MILL) ;
 N2 T2 ;
 N3 G00 G90 G54 X3.25 Y0 S3600 (S4000) M03 ;
 *N4 G43 H01 Z0.1 ;                                            M08
 *N5 G01 Z0 F5.5 (F5.) ;            F 110%    S 90%                  POS 15
 *N6 X-3.25 ;                                                                  ADD PASS
 N7 G00 Z0.1 M09 ;
 N8 G53 Z0 ;
 N9 M30 ;
 ;
 T2 M06 (.375 END MILL) ;
 T3 ;
 G00 G90 G54 X0. Y1.25 S5000 M03 ;
 G43 H02 Z1. M08 ;
 G01 Z-0.3 F50. ;
 G01 G41 D02 X-0.5 F50. ;
 G03 X0. Y0.75 R0.5 F5. ;
 S9800 G01 X1.94 (OK) ;
 G02 X2. Y0.69 R0.06 ;
 G01 Y-0.69 ;
 G02 X1.94 Y-0.75 R0.06 ;
 G01 X-1.94 ;
 G02 X-2. Y-0.69 R0.06 ;
 G01 Y0.69 ;
 G02 X-1.94 Y0.75 R0.06 ;
 G01 X0. ;
 G03 X0.5 Y1.25 R0.5 ;
 G01 G40 X0. ;
 G00 Z0.1 ;
 X1. Y0 ;

EDITOR HELP (PRESS F1 TO NAVIGATE)   CLIPBOARD
                                      008998 ;
 HELP-HOW TO USE THE EDITOR
 ---
 CONTENTS
 1) General
 2) Dual-screen design
 3) Pop-up menu
 4) Context sensitive help
 ---
 1) General
 The HAAS editor adds functionality
```
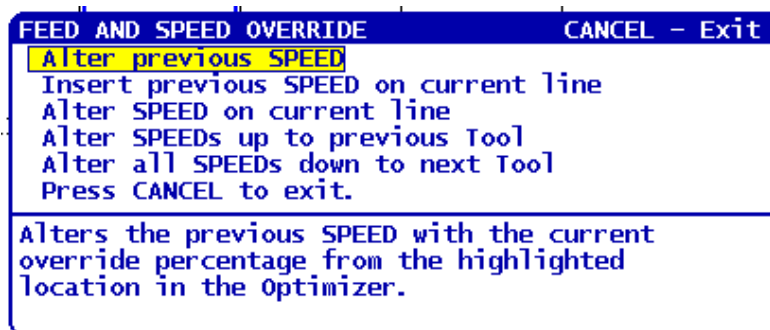
Press the Enter key. The following pop up appears.

```
COOLANT ON/OFF                        CANCEL - Exit
  Insert Coolant Command
  Press CANCEL to exit.

This will insert a Coolant Command
```

Pressing the Enter key again and M08 will be entered on line N5. Highlighting POS 15 notes that the Programmable Coolant Position was changed on N6. Pressing the Enter key gives the following pop up.

```
COOLANT POSITION                      CANCEL - Exit
  Alter P-Cool Position
  Press CANCEL to exit.

Alters the previous PCool position in the Tool
Offset page with the new position from the
highlighted location in the Optimizer.
```
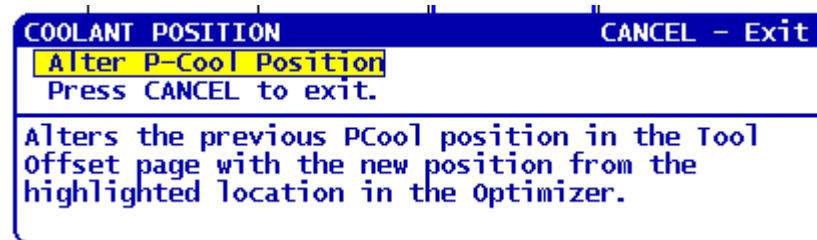
Highlighting Alter P-Cool Position and pressing Enter gives:

```
COOLANT POSITION

  The default Coolant Position is POS 15 this can
  be changed by modifying the Coolant
  Position on the input line.

  Press ALTER to accept Coolant Position.
```
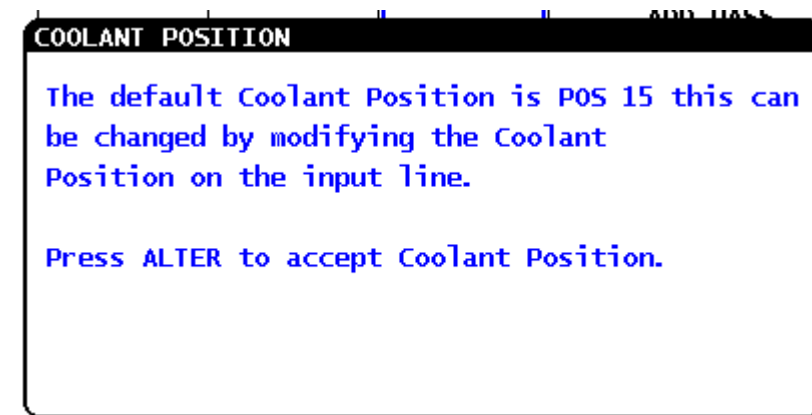
Pressing ALTER will change the Coolant Position in offsets page to 15.
Next, cursor to the note column and highlight the ADD PASS.

Pressing Enter will add the note to the Program:

```
O00010 (CLASS) ;
N1 T1 M06 (2 INCH FACE MILL) ;
N2 T2 ;
N3 G00 G90 G54 X3.25 Y0 S3600 (S4000) M03 ;
N4 G43 H01 Z0.1 ;
*N5 G01 Z0 F5.5 (F5.) ;
*N6 X-3.25 ;
*N7 G00 Z0.1 M09 (ADD PASS) ▌
N8 G53 Z0 ;
N9 M30 ;
```
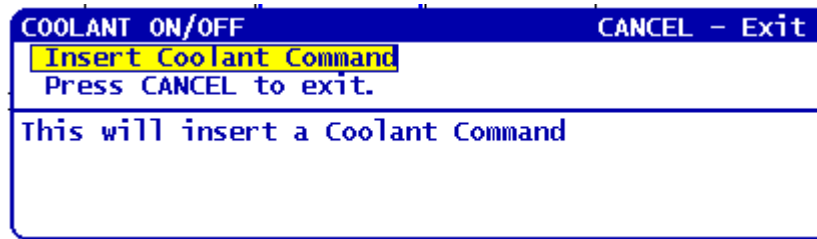
The advantage of using the program optimizer is that any changes that have been made using the override keys, the coolant position keys, coolant on or off keys, or any notes will be stored. These changes may then be made permanent into the original program with relative ease.

Haas Advanced Tool Life Management

Advanced Tool Management allows several of the same tools to be loaded in the machine. They are automatically called up when the life of one of them is ended. For example, when the tool life of T1 is used up, the machine will automatically index to the next identical or back up tool. The machine will not switch to the new tool in the middle of a program. When the limit of a particular tool is reached the Tool Life Management will not take effect until the next time the program is run from the beginning.

Advanced Tool Management is accessed by pressing the **CURNT COMDS** key found under the **DISPLAY** section of the keyboard. Press the PAGE UP key until the following screen comes up. It is divided up into three different windows.

Active Tool Label

Tool Life Limits Window

Tool Group Window

```
ADVANCED TOOL MANAGEMENT                           TOOL    1 IN SPINDLE
(TOOL GROUP)                       PRESS F4 TO CHANGE ACTIVE WINDOW
GROUP     1001    GROUPS   1 of  1      USAGE:      0
<PREVIOUS>  <NEXT>  <ADD>  <DELETE>     HOLES:      0
        <RENAME>         <SEARCH>   FEED TIME:      0
GROUP USAGE:   IN ORDER             TOTALTIME:      0
DESCRIPTION:                        TOOL LOAD:      0  TL ACTION:    ALARM
                                      WARNING:      0

TOOL#  EXP  LIFE
    0             CRNT PKT      H-CODE      D-CODE      FLUTES
    0                 0
    0             HOLES     FEED TIME TOTAL TIME    USAGE       LOAD
    0
    0
    0




Enter a four digit number and press WRITE/ENTER to create a new tool group.
```

Tool Data Window

After calling up the Advanced Tool Management page a TOOL GROUP must be set up. Highlight in yellow **<ADD>** as above. Key in a four digit number and press the ENTER key. In the above example a **TOOL GROUP 1000** was set up. Using the down then right arrow gives different options for the order that the tools are used: **IN ORDER, NEWEST, or OLDEST**. **IN ORDER** was selected using the right arrow cursor key. Next the down arrow cursor key is depressed to the DESCRIPTION line. (¼ INCH DRILL) was keyed in and entered. See below:

```
GROUP     1000    GROUPS   2 of  2
<PREVIOUS>   <NEXT>  <ADD>  <DELETE>
        <RENAME>          <SEARCH>
GROUP USAGE:   IN ORDER
DESCRIPTION:   1/4 INCH DRILL
```

The **1000 Group** was set up identified as **¼ INCH DRILL**. In G-Code a particular tool group is denoted by using **T1000** instead of **T1.** See the G-Code below

T1000 M6 (T1) (1/4 inch drill)
G90 G54 G00 X1.0 Y1.0 (XY Start Point)
S400 M03
G43 **H1000** Z1.0 M08 (H code same as the group number)

Note the **H value must also call out the group number**. If any **D values** are used in the program relating to tool group 1000 they **must also use the group number**.

Note: **Setting number 15** H and T Code Agreement must be set to **OFF** or Advanced Tool Management will not work. With setting 15 set to ON alarm 332 H and T Not Matched alarm will come up.

To navigate the three different windows press the **F4** key. The active window border is identified by a heavier border and the yellow highlighting background will migrate to the active window.

Several ways may be used to describe the tool life of a particular tool. Pressing **F4** will make the **Tool Life Limits Window** active below:

```
                         TOOL  11 IN SPINDLE
             PRESS F4 TO CHANGE ACTIVE WINDOW
        USAGE:        0
        HOLES:        0
   FEED TIME:         0
   TOTALTIME:         0
   TOOL LOAD:         0   TL ACTION:      ALARM
     WARNING:        50
```

To set a particular limit the line must be highlighted in yellow.

Usage: The total times a tool is used (called up with a M6 code)
Holes: The total number of holes allowed for a drill to cut
Feed Time: Total time in minutes a tool is allowed to be used in a feed (must be an integer).
Total Time: Total time in minutes a tool is used (must be an integer).
Tool Load: The maximum load on the spindle for a tool.
TL Action: Action when a tool has reached the end of its life. Right cursor arrow gives options.
    Alarm, Feed Hold, Beep, Auto Feed, Next Tool,
 Warning: The minimum value a tool will appear as having low life (highlighted in yellow) in the Tool Life screen.

Below gives an Advanced Tool Management screen where two tools have already expired and T3 is the active too which is doing the drilling. Note in the upper left hand corner the Tool Group Window: **Group 1000** was set up as group **¼ DRILL**. The manner in which the tools were to be used was **IN ORDER.** In the Tool Life Limits Window in the right upper corner the limit for the drill is set to **6 HOLES**.
The black bar on top gives that Tool 3 is in the Spindle.

```
ADVANCED TOOL MANAGEMENT                        TOOL   3 IN SPINDLE
(TOOL DATA)                          PRESS F4 TO CHANGE ACTIVE WINDOW
GROUP    1000    GROUPS   1 of  1      USAGE:      0
<PREVIOUS>  <NEXT>  <ADD>  <DELETE>    HOLES:      6
       <RENAME>        <SEARCH>       FEED TIME:   0
GROUP USAGE:   IN ORDER               TOTALTIME:   0
DESCRIPTION:   1/4 DRILL              TOOL LOAD:   0   TL ACTION:   ALARM
                                      WARNING:     0

TOOL#  EXP  LIFE
                       CRNT PKT    H-CODE     D-CODE     FLUTES
  1           0%          1          3          3          2
  2           0%
  3          17%        HOLES    FEED TIME  TOTAL TIME   USAGE     LOAD
  4         100%          5        0:00:04    0:00:29      2        11
  0
  0
```

The Tool Data screen is the active screen above. Highlighted in yellow Tool 3 has 17% life left. Cursoring to the right gives:

   TOOL #: Gives the tool number. Pressing Origin key and it will zero out all the data in the window for the respective tool.

   EXP (Expire) is used to obsolete a tool. A tool may manually made obsolete by keying in an asterisk (*) and then pressing Enter key.

   CRNT PKT: The pocket the highlighted tool is in.

   H-CODE: Gives the H number that is used when tool 3 is used. This may be changed to a different number

   D-CODE: Gives the D number that is used when tool 3 is used. This may be changed.

   FLUTES: The number of flutes the tool has.

   HOLES: The current number of holes the drill has drilled.

   FEED TIME /TOTAL TIME: Gives the time the tools has been in feed mode and total time respective.

   USAGE: The number of times the tool has been called up using an M6 command

   LOAD: Max load the spindle has seen running the tool

After new tools have been replaced with new tools it is necessary to go back in the Advanced Tool Management page in Current Commands. F4 to the Tool Data Page highlight the TOOL# and press the **ORIGIN** key. This will zero out the data of the respective tool in the Tool Data Page.

Another way to visually manage Tool Life is in the **Tool Life page** in Current Commands. See below.
Highlighted in blue is the tool that is in the spindle. Note on the right LIFE column T3 has 17% life left.

Note that all the tools with a blue asterisk are the tools being used in Tool Management Group ID 1000
given in the lower right box. After Tool 3 expires Tool 4 will be called up. After all the tools in the Tool
Management have expired an **alarm 471** will come up (**OUT OF TOOLS**). To continue, the spent tools will
have to be replaced and toughed off. Also they will have to be set back to zero usage in the Tool Data
Window on the Advanced Tool Management Page using the ORIGIN key.

```
TOOL LIFE                                        TOOL    3 IN SPINDLE

  TOOL    TOTAL TIME   FEED TIME   USAGE   ALARM   LIFE
   1       0:01:06     0:00:09       3       0    *   0%
   2       0:00:52     0:00:09       2       0    *   0%
   3       0:00:29     0:00:04       2       0    *  17%
   4       0:00:00     0:00:00       0       0    * 100%
   5       0:31:28     0:04:30      21       0      100%
   6       0:07:18     0:03:00      22       0      100%
   7       0:20:04     0:00:00       0       0      100%
   8       0:00:13     0:00:00       0       2      100%
   9       0:01:33     0:01:13       0       0      100%
  10       0:31:06     0:14:15      22       0      100%

                                            TOOL MANAGEMENT

                                            GROUP ID:    1000

                                            TOOL   EXPIRED   LIFE
                                            ----   -------   ----
                                              1                0%
                                              2                0%
                                              3               17%
                                              4              100%
                                              0
                                              0

 * Advanced tool management value.

   COOLANT      H(LENGTH)             D(DIA)                    ACTUAL
   POSITION  GEOMETRY    WEAR    GEOMETRY    WEAR    FLUTES    DIAMETER
      0       4.3174    0.0000    0.7496    0.0000     2       0.0000
```

Program example using Advanced Tool Life Management: Note the T1000 and H1000 call up the tool life
management group 1000 for tools and offset values.

O00002 (ADVANCED TOOL MGMT)
T1000 M06 (.09 DRILL)
G00 G90 G54 X1. Y1.
S400 M03
G43 **H1000** Z1. M08
G81 Z-0.32 R0.1 F200.
G91 X1. L4
G00 G90 G80 Z0.1 M09
G00 G53 Z0
M30

Haas Fixture Clamp Input (Mill Parameter 738)

In high production environments it is easy for the machine operator to forget to clamp a fixture or vise before the cycle start button is depressed. During the normal work routine the operator may be interrupted or distracted and fails to clamp the vise or collet. When this happens the tool is usually destroyed and the vise jaws or fixture become damaged and need replacement. To prevent this from happening Haas provides a Fixture Clamp Input. It is available on mills with software version of 15.05A or higher.

Open and close clamp switches are required. If a hydraulic vise is used a pressure switch may put on the hose going to the vise. A similar arrangement may be used If a pneumatic clamping device is used. Limit or proximity switches may also be used in fixture clamping.

The clamp switch is connected with a cable to a discreet input connector on the I/O Board. The machine interprets an open switch as a fault indicating the fixture is in the unclamped position during the machine cycle. Parameter 738 indicates the input number on the I/O Board.

If the spindle is on and the Haas mill controls sees that the part is unclamped (open circuit on the I/O Board) **Alarm 793 Fixture Clamp Failure** is generated which stops the spindle and program. This not only works at the beginning of the program but also if the part becomes unclamped in the middle of the cycle for some reason. The alarm comes on only when the spindle is commanded on. This means that a part may be unclamped in the middle of the cycle as long as the spindle is not commanded on without generating an alarm.

See Haas ES0670 rev A 10/08 titled **Fixture Clamp Input** which is included on the next page for details on installation.

Fixture Clamp Input

Parameter 738, Fixture Clamp Input, stops the spindle, stops the program and generates Alarm 973 if a fixture achieves an unclamped position. It is available for mills with software version 15.05A or later.

Operation of this parameter requires that the customer provide the fixture, clamp switch and cable. The clamp switch must be attached to the fixture and the cable must be routed from the switch to an open port on the I/O Board.

> **NOTE:** The clamp switch and cable are provided if the machine is shipped with the Robot Ready Interface.

To activate Parameter 738, press PARAM DGNOS, type 738 and press the down arrow to highlight 738 FIXTURE CLAMP INPUT. Enter the I/O Board input number (01-63).

To determine the input number, select a free I/O Board connector, short a signal to ground and observe the diagnostic screen to see which input changes to zero. P1 to P22 are on the first Discrete Inputs page with the numbering starting in the upper left hand corner with 00 (deactivates the feature) and ending at the bottom of the second column with 31. P23-28 are on the second Discrete Inputs page, which begins with 32 in the upper left hand corner and ends at the bottom of the second column with 63.

> **NOTE:** An open circuit on the I/O Board indicates a fault in diagnostics by displaying a "1" (i.e., fixture is in an unclamped condition).

Clamp Switch

If the switch has mechanical contacts, pinouts are not important. However, if a 2-wire or 3-wire active switch is used, the pinouts are important. The circuit runs at 12VDC, so any active switches must work at this voltage. Switches pull signal to ground so active types must be Negative-Positive-Negative (NPN). If more than one clamp switch is used, all but one must be mechanical, and they must be placed in series to the selected I/O Board input. If an alarm is generated, further troubleshooting is required to determine which clamp switch is in the unclamped position.

I/O Board Input

Typical pinouts starting with the bottom of the selected connector and working up are: shield, ground, signal, more signals if more than 3 pins and +12V. P22, P23, P28 have two ground pins before the signal pins. Only P2A,P10,P15,P21,P23,P25,P27 and P28 have the +12V pin.

Alarms

A CNC program is stopped by an alarm when the fixture is unclamped and the spindle is commanded on. That means a program can command a fixture to unclamp without generating an alarm as long as the spindle is not turning.

If the fixture is unclamped and 738 FIXTURE CLAMP INPUT is set to a non-zero value, then running the spindle generates Alarm 973, Fixture Clamp Failure, and stops the program and the spindle.

Advanced settings

Setting 53 – Jog w/o Zero Return

> When a Haas vertical machining is powered up using the power on key the machine has the requirement to be taken home in the X, Y and Z axes. This is accomplished by first depressing the orange RESET key which turns on the servo motors. Then the POWER UP blue key beside it must depressed to take the machine to Home. At home the X, Y and Z machine coordinates are set to zero. First the Z goes to home, then the X and Y go to Home position.
>
> If a tool is engaged into a part when it was shut off a Z positive move may damage the tool or part at start up. Setting 53 allows a jog movement to disengage the tool from a part before the obligatory machine home process.   This however puts the machine in a dangerous situation as various axes could be run into the mechanical stop and damage the machine. **After** the tool is jogged off a part **using setting 53 it is obligatory that the machine be zero returned in all axes to prevent damage to the machine.**

Setting 142 – Offset Change Tolerance

> This offset will create a warning pop up which will notify that an offset change more than the value entered in this setting is issued. The message "Greater than Setting 142! Accept(Y/N)" will appear. If a Y is entered the control will change the offset amount by the amount specified. If an N is entered than the offset change will be ignored.   Offset 142 will prevent inadvertent large offset changes from   being made which may possibly cause a crash.

Setting 156 – Save Offset with Program

> When this setting is set to On the control will save the offsets when the program is saved. The offset program is saved as O999999. In the program file it is saved after the M30 and before the %. When this program is uploaded to the Haas control from a UBS device a prompt will appear "LOAD OFFSETS? (Y/N)".

Below is an example of a Tool Offset file. First Tool Length data along with coolant position and wear are stored from H1 to H200. Next Work Offset data is stored G52-G59. Then extra work offset data is stored as G110 corresponding to G154 P1 to G154 P99. At the end data on the location of tools in pocket is stored titled (POCKETS AND TOOLS). When loading the offsets the (POCKETS AND TOOLS) data is only loaded if setting 155 is set to ON (see next section).

```
M30
 O999999
(TOOL OFFSET)
N000 V-20375
(DIMENSIONING= INCH)
(LENGTH AND WEAR)
(  CLNT_P  L_GEOM  L_WEAR  DIA_GEOM  D_WEAR  FLUTES)
H01  C19   L6.2618 W0.0000 D1.9931 E0.0000 F6
H02  C08   L3.4035 W0.0000 D0.5001 E0.0000 F2
H03  C08   L3.7344 W0.0000 D0.7505 E0.0000 F2
H04  C15   L7.2445 W0.0000 D0.0000 E0.0000 F2
H05  C08   L2.9570 W0.0000 D0.0000 E0.0300 F2
H06  C11   L4.4207 W0.0000 D0.0000 E-0.0300 F2
H07  C14   L5.1906 W0.0000 D0.0500 E0.0000 F1
H08  C00   L0.0000 W0.0000 D0.0000 E0.0000 F2
H09  C11   L3.9950 W0.0000 D0.0000 E0.0000 F2
H10  C17   L5.9283 W0.0000 D0.0000 E0.0000 F2
(H11-H199)
H200 C00   L0.0000 W0.0000 D0.0000 E0.0000 F2
(WORK ZERO OFFSET)
(    X       Y       Z       A       B       C )
G52   X0.0000 Y0.0000 Z0.0000 A0.000 B0.000 C0.000
G54   X-13.9574 Y-8.5363 Z-17.1033 A0.000 B0.000 C0.000
G55   X-16.9583 Y-8.5367 Z0.0000 A0.000 B0.000 C0.000
G56   X-15.7454 Y-10.0378 Z0.0000 A0.000 B0.000 C0.000
G57   X-18.1170 Y-8.8107 Z-17.2540 A0.000 B0.000 C0.000
G58   X-15.8496 Y-9.8282 Z-17.2538 A0.000 B0.000 C0.000
G59   X-15.7986 Y-11.0222 Z-17.7339 A0.000 B0.000 C0.000
G110   X-14.5201 Y0.0000 Z0.0000 A0.000 B0.000 C0.000
(G111-G154 P98)
```

```
G154 P99 X0.0000  Y0.0000  Z0.0000  A0.000  B0.000  C0.000
G92    X0.0000  Y0.0000  Z0.0000  A0.000  B0.000  C0.000
(POCKETS & TOOLS)
T01  P1   L32
T02  P19  L32
T03  P20  L32
T04  P24  L72
T05  P4   L32
T06  P9   L32
T07  P11  L72
T08  P14  L32
T09  P17  L32
T10  P15  L32
T11  P6   L32
T12  P5   L32
T13  P2   L32
T14  P10  L32
T15  P16  L32
T16  P3   L32
T17  P12  L32
T18  P7   L32
T19  P23  L32
T20  P21  L32
T21  P25  L32
T22  P8   L32
T23  P22  L32
T24  P18  L32
T25  P13  L32
%
```

Setting 155 – Load Pocket Tables

If this setting is ON the Pocket Tool table will be altered to reflect the data in an Offset file when it is loaded. Normally setting 155 is set to OFF.   When the machine is turned on setting 155 is set to OFF and is the default. So normally when an offset file is loaded into the controls just information on tool and work offsets is loaded. According to Haas this setting 55 should only be used when software has been upgraded, the memory cleared or re-initialized. In the side-mounted tool changer tool positions in particular pockets are constantly changing with operation of the machine. With setting 155 set to ON when offsets are loaded up the location of current tools in the turret will be lost. Use of the setting may have an application in set up situations where tools have been loaded up after the tool pocket table has been set to default values. Pressing the Origin key will reset the Pocket Tool Table to default values. At default Tool #1 is in the spindle, Tool 2 is in pocket position #1, Tool 3 is in pocket position #2, etc.

Below is an example of a Pockets & Tools file. Interpretation of the file is somewhat counter intuitive. The T's represent pocket positions while the P represent Tool numbers. L72 represents a heavy tool and L32 represents a regular tool. T01 represents a tool in the spindle. T02 represents a pocket position #1. Per table below Tool 19 (P19) would be in pocket position #1. In the third line Tool 20 (P20) would be in pocket position #2 (T03).   The spindle probe is T24. In line four we find T24 (P24) in pocket position #3 (T04) noted as a heavy tool (L72).

(POCKETS & TOOLS)
T01 P1   L32
T02 P19 L32
T03 P20 L32
T04 P24 L72
T05 P4   L32
T06 P9   L32
T07 P11 L72
T08 P14 L32
T09 P17 L32
T10 P15 L32
T11 P6   L32
T12 P5   L32
T13 P2   L32
T14 P10 L32
T15 P16 L32
T16 P3   L32
T17 P12 L32
T18 P7   L32
T19 P23 L32
T20 P21 L32
T21 P25 L32
T22 P8   L32
T23 P22 L32
T24 P18 L32
T25 P13 L32

Tool Length Offset and Cutter Radius Compensation Techniques

Tool Length Offset Compensation

Tool length compensation allows the programmer to not worry about the length of the tool and program to the top surface of the part. Normally the top surface of the part is set as Z zero.
The following command turns on tool compensation for a particular tool. In this instance Tool #1.

<div align="center">

G43 H01 Z.1

</div>

The above command (normally given with the machine in the rapid mode, G00) will rapid to .100 inch above the top face of the part using Tool Length Offset 1. The program would use the value in line 1 and column labeled H Geometry, 3.5000.

```
                    PROGRAM TOOL OFFSETS
  IPS ON      COOLANT        H(LENGTH)           D(DIA)
    TOOL      POSITION    GEOMETRY    WEAR    GEOMETRY    WEAR
  1 SPINDLE      0          3.5000    0.       0.5000     0.
    2            0          5.1000    0.       0.1250     0.
    3            0          6.2000    0.       0.2500     0.
    4            0          5.0000    0.       0.0890     0.
    5            0          5.0000    0.       0.0910     0.
    6            0          5.0000    0.       0.1250     0.
    7            0          5.0000    0.       0.        0.
    8            0          5.0000    0.       0.        0.
    9            0        -10.0000    0.       0.        0.
   10            0          0.        0.       0.        0.
  ENTER A VALUE
```

During a set up if T1 was a ½" end mill and a close tolerance .200 +/-.0005" depth of pocket needed to be held the prudent machinist would add .01" in the WEAR column of T1. The part would be run and the depth of the pocket measure in the machine. If the value was .192 the cutter needs to go .008 deeper.   So -.008 would be added to the Wear of the tool which would leave it at .002". The machinist prevents from possibly scraping out the first piece by adding a small amount to the wear offset.

Another situation may come up where two close tolerance depth features (+/-.0005) are created with the same tool #1. If both tolerances cannot be held with one length offset then a **secondary offset** may be used. Pick an offset that is not being used such as 10 above. Put the same H Geometry value in 10 as in 1. Also put the same Wear in 10 that was in 1. Then either add or subtract the amount to make the second feature so it will be at nominal value. The program needs to be edited to reflect using a secondary offset. In the program after the first feature is completed and the Z is above the top face of part: (G43 H10 Z.1.) is inserted for the second offset.

Cutter Radius Compensation Sizing

Cutter radius compensation allows the programmer to forget about the radius of the tool and essentially program the features on the print. The computer inside the controls figures which coordinate values need to be changed to reflect the radius of the cutter. The following command turns cutter compensation on:

G41 X.0 D1.

The D value is stored in the Tool Offset page under the D column for each tool. G41 command is cutter compensation left, G42 command is cutter compensation right. Most applications on CNC mills use G41 which is climb cutting. It gives better finish and tool life than G42 cutter compensation right. G42 is used for conventional cutting.

Cutter compensation not only compensates tool location for the radius of the tool it also may be used for sizing or dimensional control. The following illustration shows a ½ inch cutter cutting off the end of a part to take it to size. If cutter compensation was not used the only way to size the 4.000 length was to change the stop location on the left side of the part or edit the program.
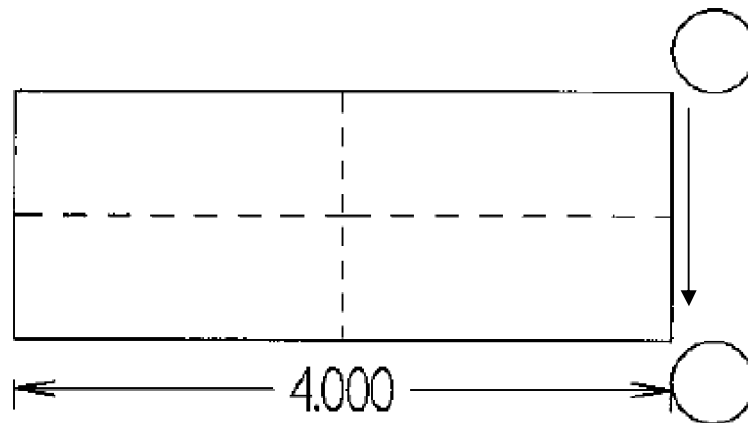
Using cutter compensation all that needs to be changed is the size of the radius or diameter value in the D value of the Tool Offset screen. If the D value is smaller the machine will compensate by taking more material off when it cuts. If the D value is larger the machine will move further away from the cutting edge therefore leaving more material on.

D values on Haas Controls may be designated as diameters or radii in **setting #40**. For example the first part is cut and the part measures 4.020". .020" more material needs to be taken of the end of the part to take it to the nominal dimension of 4.000. If the Haas machine is set up where D is **radial** value a negative (**-.020)** would be added in the D Wear column. If the machine is set up in **diameter mode two times** the value needs to added in the D column or (**-.040)**. Just the opposite if more material needs to be left. Positive values are added to the D value.
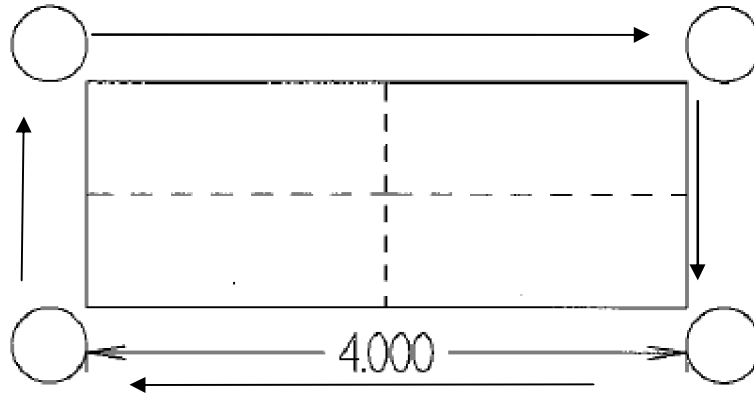
In the above example the part is cut and the 4.000" measures 3.980". Now the part is being cut on two sides so .01" needs to be added to each side. If the D is in diameters (**+.02"**) is added to the D Wear. If the D is in radii **(+.01")** is added to the D Wear column for tool #1.

Roughing Applications Using Cutter Compensation

One of advantages of using Cutter Compensation is that **different size tools can be used with the same program**. All that needs to be changed is the D value for the respective tools. The only limit is that the distance of the move to turn on the cutter compensation (G41 or G42 block) be larger than radius of your tool. Also their maybe limits with respect to interior radii with pocket milling. For example the part contour program below using tool compensation could be used with a 1/2" end mill or 3/8" end mill.
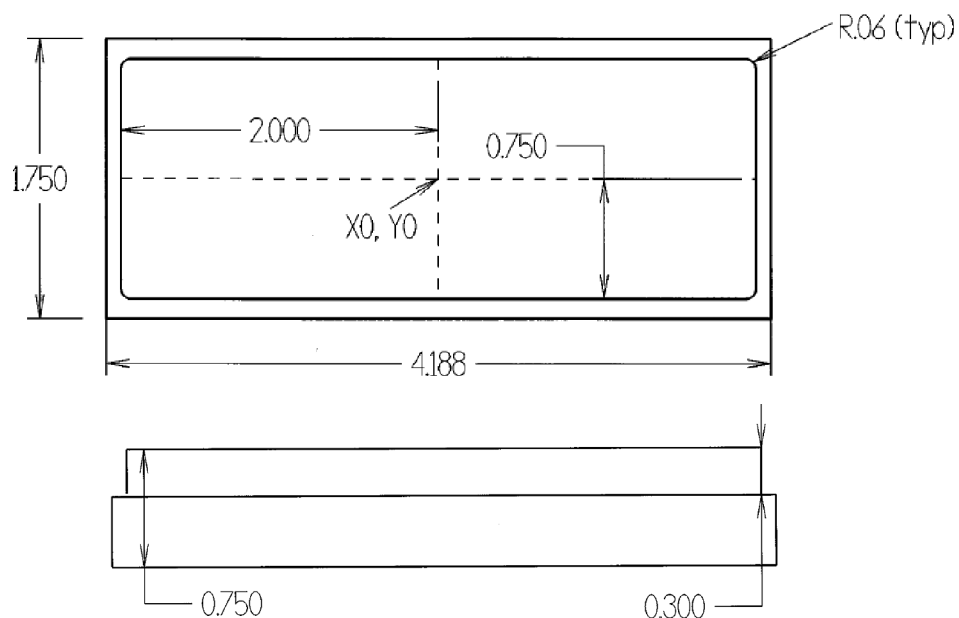
Another advantage with using tool compensation is that the **same tool path used for finishing may be used with a roughing tool**. All that needs to be done is to **manipulate the D value for the roughing tool**.

The tool paths for both will be the same. A separate roughing tool path does not need to be created. Additionally the amount left with the roughing end mill may be easily changed. Note that T2 the D value is .540, .04 larger than the size of the tool. As the machine D is set to diameters .02" will be left for the 3/8" end mill to finish. The tool path is in the subroutine N100

O00011 (CLASS)
T2 M06 (1/2" ROUGH END MILL)
T3 **(SET D=.540)**
G00 G90 G54 X0. Y1.25 S5000 M03
G43 H02 Z1. M08
G01 Z-0.3 F50.
G01 G41 D02 X-0.5 F50.
M97 P100 (TOOL PATH SUB)
M01
T3 M06 (3/8" END MILL)
(SET D3=.375)
G00 G90 G54 X0. Y1.25 S5000 M03
G43 H03 Z1. M08
G01 Z-0.30 F50.
G01 G41 D03 X-0.5 F50.
M97 P100 (TOOL PATH SUB)
M30

N100 **(TOOL PATH)**
G03 X0. Y0.75 R0.5 F50.
G01 X1.94
G02 X2. Y0.69 R0.06
G01 Y-0.69
G02 X1.94 Y-0.75 R0.06
G01 X-1.94
G02 X-2. Y-0.69 R0.06
G01 Y0.69
G02 X-1.94 Y0.75 R0.06
G01 X0.
G03 X0.5 Y1.25 R0.5
G01 G40 X0.
G00 Z0.1
G53 Z0
M99

Tool Path of Roughing and Finishing Cutters in Graphics
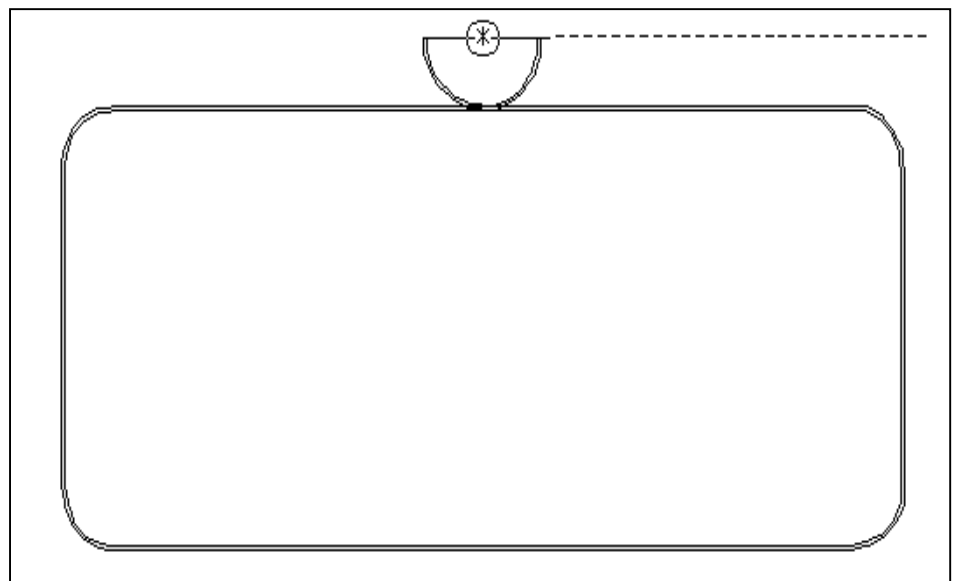
Chamfering using Tool Compensation

Using tool compensation parts may easily be chamfered using the right tools. A chamfering operation may be easily added to the program example on page 16. To create the chamfer a ¼" 90 degree point carbide N/C spot drill is selected. Then a tool operation is added using the same tool path found in **sub routine N100,** see below. The depth that the tool is set to go is Z-.080. If the tool is uncompensated the part will end up with an .08 chamfer with a nasty bur at the bottom. See Uncompensated Path illustration below. If the tool is moved .06 off of the profile as shown in the Compensated Path illustration below, a clean .02" chamfer results. To do this **D4** is set to **.06"** if the D values are set up as **radii** or **.12"** if the D values are set as **diameters**.

```
T4M06 (1/4" 90 DEGREE SPOT DRILL)
(SET D4=.120)
G00 G90 G54 X0. Y1.25 S5000 M03
G43 H04 Z1. M08
G01 Z-0.08 F50.
G01 G41 D04 X-0.5 F50.
M97 P100 (TOOL PATH SUB)
M30
```

<div style="display:flex">

Uncompensated Path
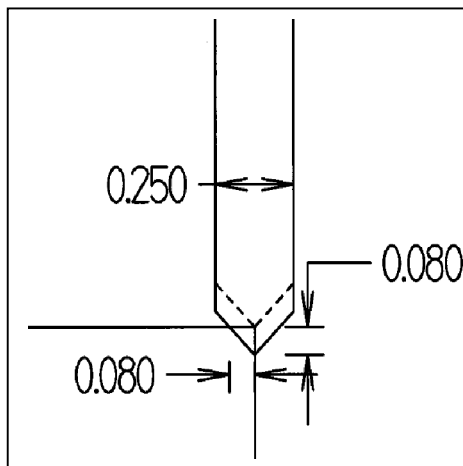
Compensated Path

</div>
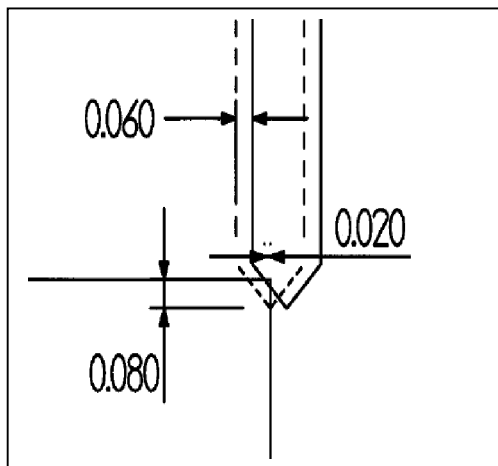
Secondary D offsets

Sometimes two different features created by the same tool cannot be held in tolerance. If it is a close tolerance one feature may be held in tolerance while the other runs out of tolerance. This may be controlled by creating both features using different D values and creating the features with cutter compensation.   An example would be two different counter bores created using the G13 function with just the I value. If each G13 had its own D value the features in question could be controlled separately:
Feature 1 (G13 I.25 F10.Z-.5 **D01**),
                Feature 2 (G13 I.3125 F10 Z-.25 **D11**)

G12, G13 Circular Pocket Milling

- É   Used for milling circular pockets
- É   G12 [D..] [Fõ] [Iõ] [Kõ] [Lõ] [Qõ] [Zõ] (Clockwise move)
- É   G13 is used for counter clockwise moves
    - D - Tool radius offset selection
    - F . Feed rate
    - I - Radius of first circle(or finished circle if no K)
    - K - Radius of finished circle (optional)
    - L - Loop count for deeper pockets (used with a G91)
    - Q - Incremental radius step (required with K)
    - Z - Depth of cut (or increment with L)

- É   G12 and G13 are Non-modal
- É   Cutter Compensation is included in this routine
- É   Use D00 to ignore tool offset
- É   Use I without K and Q for small pockets or holes
- É   When using K and Q, only K should be the radius of the desired finished pocket
- É   Position cutter in a previous block or add an X and Y to the G12/G13 line

Example of one pass milling using only the I variable.

Milling a 0.8" diameter 0.5" deep pocket using an 0.5" end mill. The picture shows the tool path for the code given. Note with G12 a clockwise rotation is created where the mill is conventional cutting.

G12 Z-0.5 I0.4 D01 F15
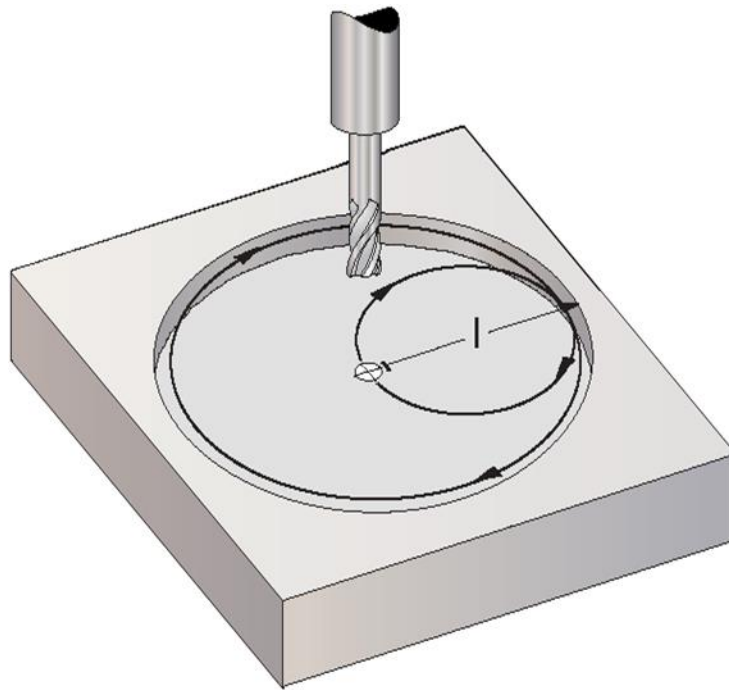
Example of multiple pass milling using I, K, and Q variables. The code is also turned into incremental positioning with G91 and an L added. So the code is repeated 3 times stepping down Z-.5 each pass.

Milling a 3.0" diameter 1.5" deep pocket using an 0.5" end mill.

O0010 ;
T1 M06 ;
G90 G54 G00 X1.0 Y1.0 ;
S1500 M03 ;
G43 Z0.1 H1 M08;
G1 Z0 F30.
G13 G91 Z-0.5 I0.3 K1.5 Q0.3 D01 F15. L3;
G90 ;
G00 Z0.1 M09 ;
G28 G91 Y0 Z0 ;
M30 ;

Note the motion of the cutter is counter clockwise using G13 instead of G12. The radius of the first pass is determined by I.3. First the ½ end mill plunges in creating ½ hole, then the first cut creates a .6 diameter pocket with I .3. The second cut adds Q.3 to the I.3 to equal a .6 radius cut out to a 1.2 diameter. Additional cuts are added with the Q.3 step over until the final K radius or finish pass is completed. Then the whole process is repeated two more times stepping down incrementally the Z-.5. The final pocket depth equals 3 x .5 to give 1.5. The above illustration just shows the initial step of three.
Note the end mill is positioned at the center of the pocket and fed down to **Z0** the top face of the part.

The above illustrations and some text taken from Haas Microsoft Power Point titled "Unique G- Codes"

Arc On, Arc Off with Tool Compensation

When finish machining a part a rule of thumb is always leave equal amounts of stock on all features in milling or turning. The reason for this is because of tool pressure. Tool pressure is the amount of force that is generated on a tool when it is cutting. A deeper cut creates more force on the tool than a smaller cut. Tool pressure has a tendency to deflect the tool away from the part. The larger the tool pressure the larger the deflection of the tool. If there is more stock on one feature than another the tool is deflected further away from the part creating possible out of tolerance conditions.   Another aspect is that once a tool becomes dull the larger the tool deflection. Out of tolerance conditions may develop as tools becomes duller where they may not be seen with new sharp tools. Because of tool deflection it is prudent to evenly distribute tool pressure throughout a single machining operation. To accomplish this, a roughing pass may be used so that the same amount of material is left for the finish pass.

When finished machining a contour or pocket, witness marks or slight gouges occur when the tool ends at the start position. At the end position the tool is not cutting material, no tool pressure and therefore there is no tool deflection.   Because it is not deflected it will cut off more at the end point than when travelling around the part cutting a certain amount of stock off. Below the tool ends up where it starts when using G41 cutter compensation for contour cutting the outside of the part below.
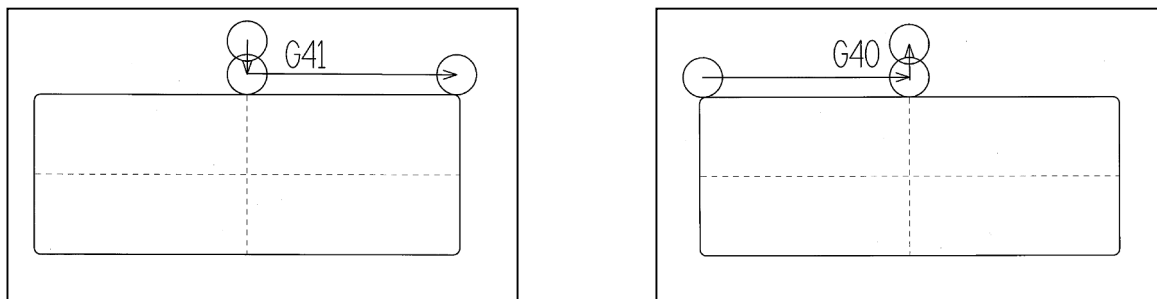
Using an arc on the part and an arc off minimizes this effect. When the tool arcs on to the part it slowly cuts off more and more material until the full depth of the stock is reached.   The opposite happened when it arcs off the part.   Less and less material is removed until the end position is reached. The approach and retract movements are smooth and less likely to leave a witness mark.

When manually programming an arc on and arc off the radius of the arc must be greater than the radius of the cutter.   It is easier to program a 90 degree arc.   On the next page a ½ inch arc and arc off is programmed starting ½" away from the part.

Example of Arc On Arc Off Using G41 Cutter Compensation
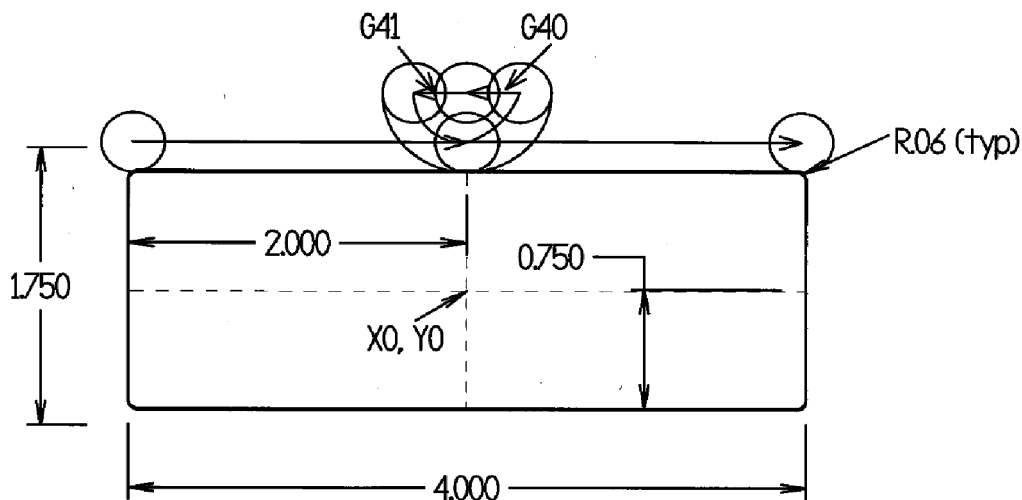
In this example the tool starts ½ " away from the edge of the finished part. It turns on and off tool compensation with a ½" move parallel to the surface of the part. It creates a ½ " radius when it arcs on and off. It forms a 90 degree arc. The arrows on G41 indicate the moves when cutter compensation is turned on. It then arcs onto the contour part. The tool then machines the finish profile of the part till it gets to the position it started from and arcs off the part. The G40 arrow indicates where the cutter compensation is turned off.

```
T2 M06 (.375 END MILL)
G00 G90 G54 X0. Y1.25 S5000 M03
G43 H02 Z1. M08
G01 Z-0.3 F50.
G01 G41 D02 X-0.5 F50.
G03 X0. Y0.75 R0.5 F50.
G01 X1.94
G02 X2. Y0.69 R0.06
G01 Y-0.69
G02 X1.94 Y-0.75 R0.06
G01 X-1.94
G02 X-2. Y-0.69 R0.06
G01 Y0.69
G02 X-1.94 Y0.75 R0.06
G01 X0.
G03 X0.5 Y1.25 R0.5
G01 G40 X0.
G00 Z0.1 M09
G53 Z0
M30
```

Closed Slot Exercise
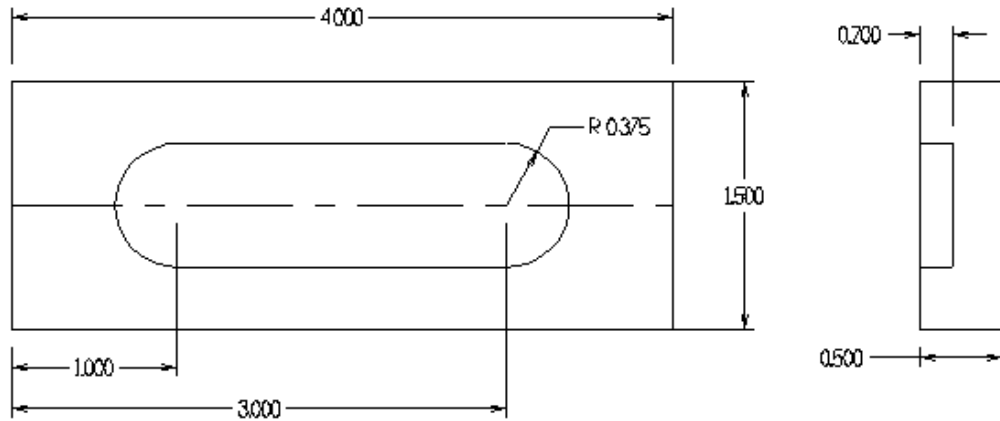
Program closed slot with a 4 flute carbide 5/8 inch end mill ramping down at 2°. Arc on and arc off with radius of .35. Speed of 1500 ft/min and .005/rev-tooth feed rate.

O03902 (SMID CLOSED SLOT)
N1 G28
N2 T17 M06
N3 G90 G54 G00 X___ Y____ S_____ M03 (ST POSITION RIGHT SID OF SLOT)
N4 G43 Z0.1 H17 M08
N5 G01 Z0 F80.
N6 X___ Z-____ F____ (RAMP DOWN TO LEFT SIDE)
N7 X____ Z_____ (RAMP DOWN TO RIGHT SIDE)
N8 X____ Z-____ (RAMP DOWN TO LEFT SIDE)
N9 X____ (FINISH BOTTOM OF SLOT)
N10 G41 X2.65 Y0.725 D17 F90. (TURN ON CUTTER COMP LEFT)
N11 G03 X___ Y_____ R0.35 (.350 RADIUS ARC ON TO RIGHT SIDE)
N12 Y_____ R____ (FINISH RADIUS RIGHT SIDE)
N13 G01 X____ (G1 TO LEFT SIDE)
N14 G03 Y_____ R_____ (FINISH RADIUS LEFT SIDE)
N15 G01 X3.
N16 G03 X_____ Y0_____ R0.35 (ARC OFF)
N17 G____ G01 X____Y_____ (CUTTER COMP OFF)
N18 G00 Z1. M09
N19 G91 G28 Z0 M05
N20 M30

Corner Rounding and Chamfering

Corner rounding and chamfering may be used to shorten and make it easier to write programs. With this utility the end points of chamfers and radii are automatically calculated. For chamfers **,C** with the length of the chamfer indicated is used. For radii a **,R** with the size of the radius indicated. The chamfering or corner rounding block may be inserted between two linear or G01 blocks. These two blocks specify a corner of the intersection of the two linear moves. The corner rounding and chamfering function interprets this as the corner and automatically calculates the start and end points of the radius or chamfer desired. The corner the rounding function will figure the radius of a circle tangent to that corner.

Below gives a print with the corner dimensions. What is desired is .06 and .5 radii on the top and .06 and .2 chamfers on the bottom.
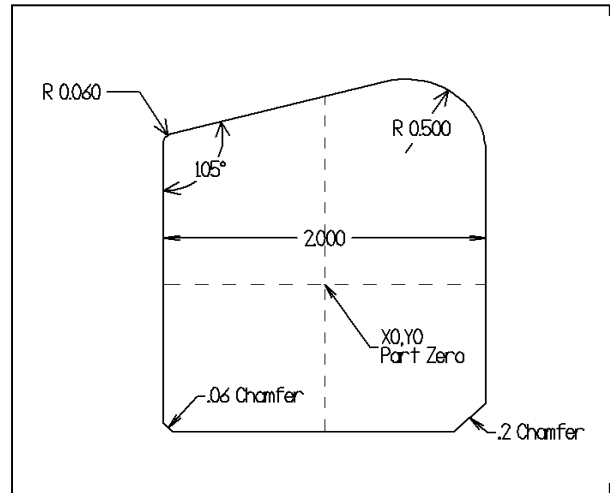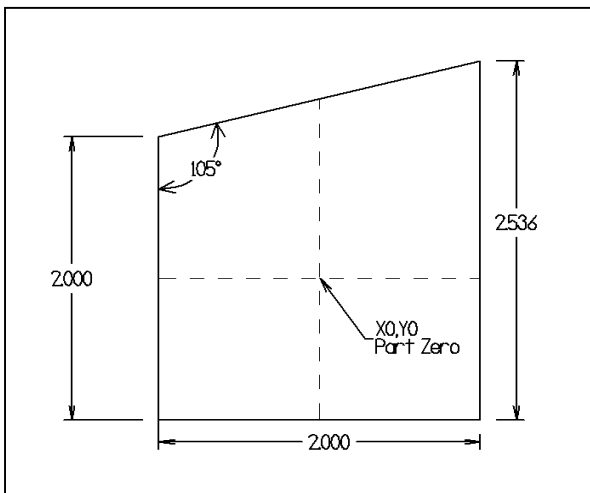
Arc on Arc off program using chamfering and corner rounding for above prints:

O00025
T2 M06 (.5 END MILL) (D=.5)
G00 G90 G54 X-1.5 Y0 S5000 M03   *(Graphics of tool path program to left)*
G43 H02 Z1. M08
G01 Z-0.5 F80.
G01 G41 D02 Y-0.5 F80.
G03 X-1. Y0 R0.5 F50.
G01 Y1. ,R0.06
G01 X1. Y1.5359 ,R0.5
G01 Y-1. ,C0.2
G01 X-1. ,C0.06
G01 Y0
G03 X-1.5 Y0.5 R0.5
G01 G40 Y0
G00 G90 Z0.1 M09
G53 Z0
M30
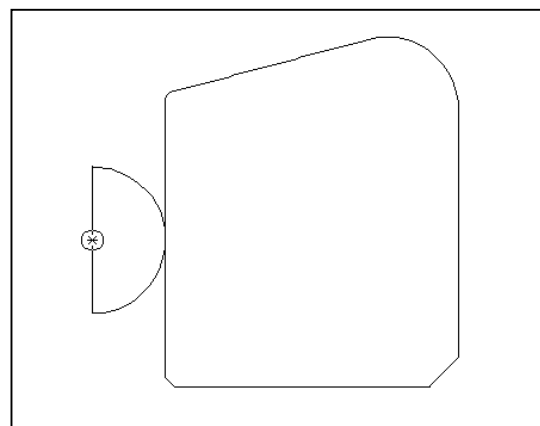
Class Exercise:
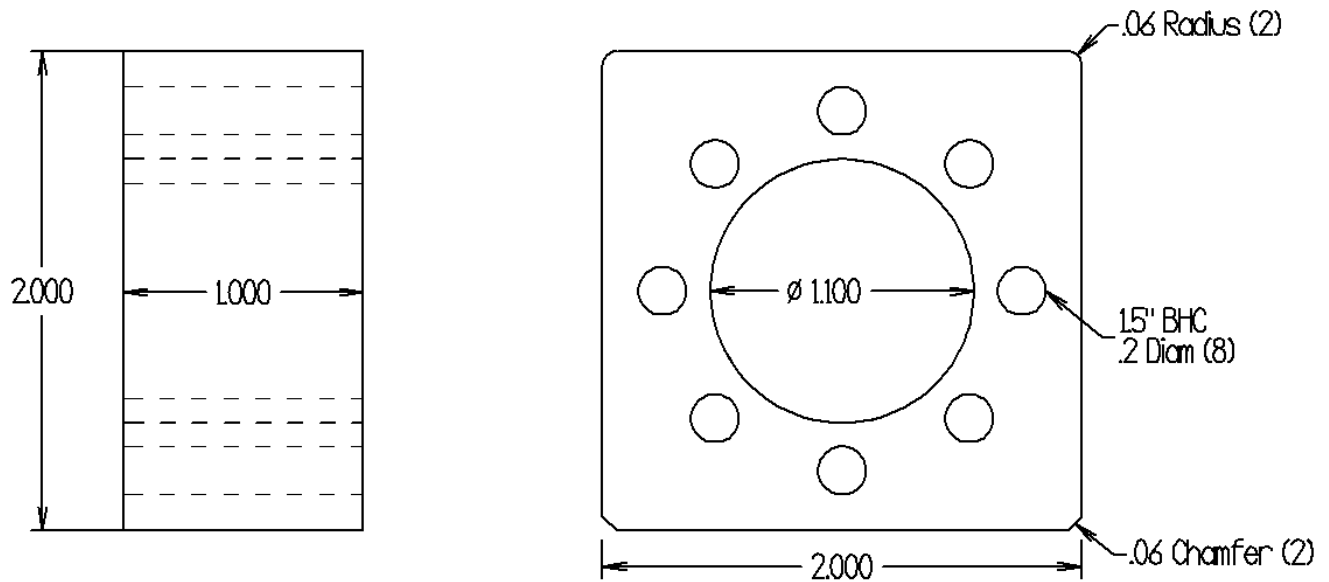
T1- 2" Face Mill (S4000, F50.)

T2- 1" Carbide Insert Drill (S 675 Ft/Min) F .005/Rev

T3- Two Flute ½ Diameter Carbide End Mill (S800 Ft/Min, F .004 In/Rev)
    Use arc on arc off to finish cut outside of part, Use G13 with G91 to finish ID Diameter to 1.100 inch step down 2 times.

T4- 3/8" Carbide Spot Drill 90 degree point, chamfer outside and inside bore, spot holes to leave .3 chamfer diameter.

T5- #7 High Speed Drill S 250 Ft/min F.004 In/Rev

T6- ¼-20 Tap

```
%
O00024 (2X2X1 CLASS VF-2 )
T1 M06 (2 INCH FACE MILL)
T2 (2 X 2.05 X 1)
G00 G90 G54 X____ Y0 S4000 M03                  (START POSITION)
G43 H01 Z0.1 M08
G01 Z0 F__.
X-____ (END  POSITION)
G00 Z0.1 M09
G53 Z0
M01
T2 M06 (1.0 INSERT DRILL)
T__  (STAGE NEXT TOOL)
G00 G90 G54 X__ Y__
S____ M03
G___ H__ Z0.1                                   (TURN ON TOOL LENGTH COMPENSATION)
G81 Z____ R0.1 F___
G00 G80 Z0.1
M05
G53 Z0 H00
M01


T2 M06 (.5 END MILL)
(D=.5)
T3
G00 G90 G54 X-1.5 Y0 S_____ M03
G43 H02 Z1. M08
G01 Z-0.5 F80.
G01 G___ D__ Y-___ F80. (TURN ON CUTTER COMP LEFT)
G03 X-___ Y0 R___ F50. (ARC ON)
G01 Y__. ,R.____ (CORNER ROUNDING)
G01 X____ Y____ ,R____(CORNER ROUNDING)
G01 X____ Y-____. ,C____ (CHAMFERING)
G01 X_____ Y-_____ ,C0.____ (CHAMFERING)
G01 Y0.
G03 X-_____ Y____ R____  (ARC OFF)
G01 G___ Y___ (TURN OFF CUTTER COMP)
G00 Z0.1
G00 X__ Y___ (POSITION TO CENTER OF PART FOR G13)
G01 Z0 F80.
G13 G__ Z-0.___ I0.___ K0.____ Q____ D___ F___. L___ (FINISH 1.1 BORE WITH G13 STEP DOWN)
G00 G90 Z0.1 M09
G53 Z0 G49
M01
```

```
T3 M06 (CHAMFER SPOT) (SET D=.___)
T4
G00 G90 G54 X-1.5 Y0 S5000 M03
G43 H03 Z1. M08
G01 Z-0.0__ F80.
G01 G41 D03 Y-___ F___. (TURN ON CUTTER COMP)
G03 X-___ Y__ R0.___ F50. (ARC ON)
G01 Y__. ,R0.___
G01 X___. Y___. ,R____
G01 X____ Y____. ,C____
G01 X___. Y-____. ,C____
G01 Y___
G03 X-____ Y0.____ R0.___
G01 G40 Y___
G00 Z0.1
X0 Y0
G13 I0.____ D03 Z-0.0_ F50. (CHAMFER BORE)
G00 Z0.1
G00 G90 G54 X0. Y0.
G81 Z-0.___ R0.1 F11. L0  (SPOT FACE BHC)
G70 I____ J___ L___
G00 G80 Z0.1 M09
G53 Z0
M01
T6 M06 (#7 DRILL)
T7
G00 G90 G54 X0. Y0.
S4000 M03
G43 H06 Z0.1 M08
G83 Z-1.15 R0.1 F11. Q0.___ L0
G70 I0.____ J__. L__
G00 G80 Z0.1 M09
G53 Z0
M01
T7 M06 (1/4-20 TAP)
T1
G00 G90 G54 X0 Y0.
S1000
G43 H07 Z0.2 M08

G84 Z-1. R0.15 F50. L0
G70 I0.____ J__ L___
G00 G80 Z0.15 M09
G94
G53 Z0
G53 Y0
M30
%
```

(\) Block Delete Application

Block Delete, also called Optional Skip, determines what happens when a line of code has a back slash mark (/). On Haas controls a Block Delete key is located on the Memory line of the Mode Keys. When it is depressed a Block Delete in black background appears on the lower right hand corner of the Control Screen. When the control of the machine sees a back slash it looks to see if the Block Delete is active. If it sees that the Block Delete function is active it ignores the whole line of code or skips to the next line of code.

Turning Coolant Off/On

For instance a line of code contains a M08 to turn on the coolant along with a Block Delete(\)
>   **/M08**

If the Block Delete is active the machine will not turn on the coolant and skip to the next line of code. In this manner whether the machine turns on the coolant or not may be controlled by the Block Delete function.

Controlling Feed and Speeds for different materials within the same program

If the same part is made of two different materials, different speeds for the different materials may be controlled within the same program using the Bock Delete function. This will save you from having two different programs of the same part made of different materials.   The following example is given for making the same part out of medium alloy steel and cast iron. The alloy steel is run a surface speed of 80 ft/min and the cast iron is run at a surface speed of 100 ft/min. The coolant is turned off for the cast iron. Both cast iron and alloy steel are fed at a rate of .005"/minute.

```
T2 M06 (1/4 DRILL)
G00 G53 Z0 (TURN ON BLOCK DELETE FOR CAST IRON)
G00 G90 G54 X0.5 Y0.5 M03
S1530
/S1220
 G43 H02 Z1.
/M08
F7.6
/ F6.1
G98 G83 Z-1.1  R.1 Q0.250
Y2.5
X3.5
Y0.5
G00 G80 Z1. M09
M05
G00 G53 Z0
M30
```

With the Block Delete off, the spindle speed is 1220 rev/min, the feed rate is 6.1 in/min with the coolant on. This corresponds to feed and speed calculated for medium alloy steel with coolant on.
When the Block Delete is on, the spindle speed becomes 1530 rev/min and the feed rate 7.6 in/min. Also the coolant is never turned on. This corresponds to the feed and speed calculated for the cast iron.

Using Block Delete for Removing Unexpected Extra Stock, Call Sub Routine

Many times material comes in with extra stock. This may be dealt with by adding another operation or just adding passes to the current program that is to be run. Using the Block Delete function extra passes may just be turned on when they are needed within the same program. See lathe program below.   In this example some parts to be machined come with .25" stock over nominal length.

```
O000010 (EX BLOCK DELETE TO ADD PASSES)
G28 (IF STOCK <.1 OVER NOMINAL OAL TURN ON BLOCK DELETE)
T101
G50 S2000
G97 S616 M03
G96 S500 M08
G00 Z.200
X3.1
/ G01 X-0.07 F.009
/G00 Z.3
/X3.1
/Z.100
/ G01 X-0.07 F.009
/G00 Z.200
/X3.1
N101 G00 Z0
G01 X-0.07 F.009
N102 G00 X3.1 Z0.1
G71 P103 Q104 D0.1 U0.02 W0.004 F0.011
N103 G00 X1.88
G01 G42 Z0
G01 X1.92
G03 X2. Z-0.04 R0.04
G01 Z-1.
G01 X2.72
G03 X2.8 Z-1.04 R0.04
G01 Z-1.5
G01 X3.1
N104 G40 X3.2
G00 X3.1 Z0.1
G70 P103 Q104 F0.007
G97 S500 M09
G00 G28 U0
G28 W0
M30
```

Block Delete may also be used for facing off extra stock on a mill.
The material comes in at .65" to .75" thick. The maximum amount to be taken off per pass is .100".
Top face of the finished part is Z0. An extra pass is added with block deletes below.
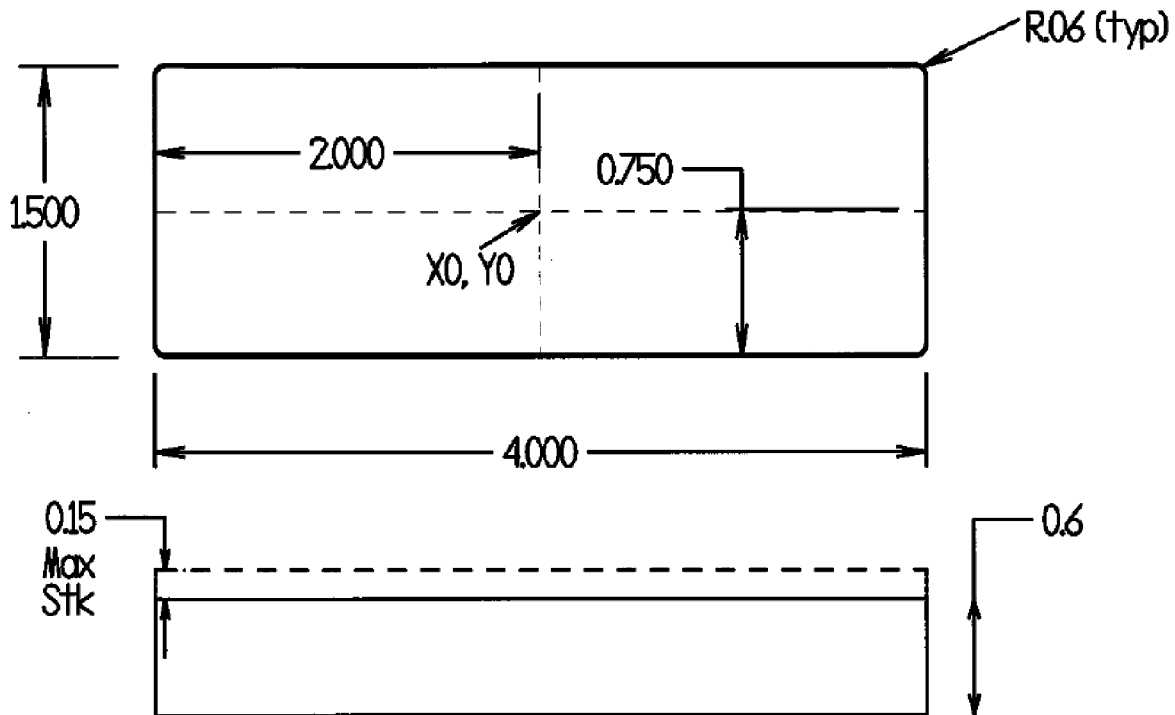
O00012 (CLASS)
N1 T1 M06 (2 INCH FACE MILL)
N2 T2 (IF STOCK LESS THAN .1 TURN BLOCK DELETE ON)
N3 G00 G90 G54 X3.25 Y0 M03 S3600
N4 G43 H01 Z0.2
/ N5 G01 Z.1 F5.5
/ N6 X-3.25
/ N7 G00 Z.2
/ N8 X3.25
N9 G01 Z0 F5.5
N10X-3.25
N11 G00 Z0.1 M09
N12 G53 Z0
N13 M30

Using Block Delete for Removing Features (Subtracting Features)

Block Delete may be used on similar parts where the only difference is that a feature has been subtracted. Instead of creating another program with subtracted features block delete may be used to subtract features within the same program.

Part 101
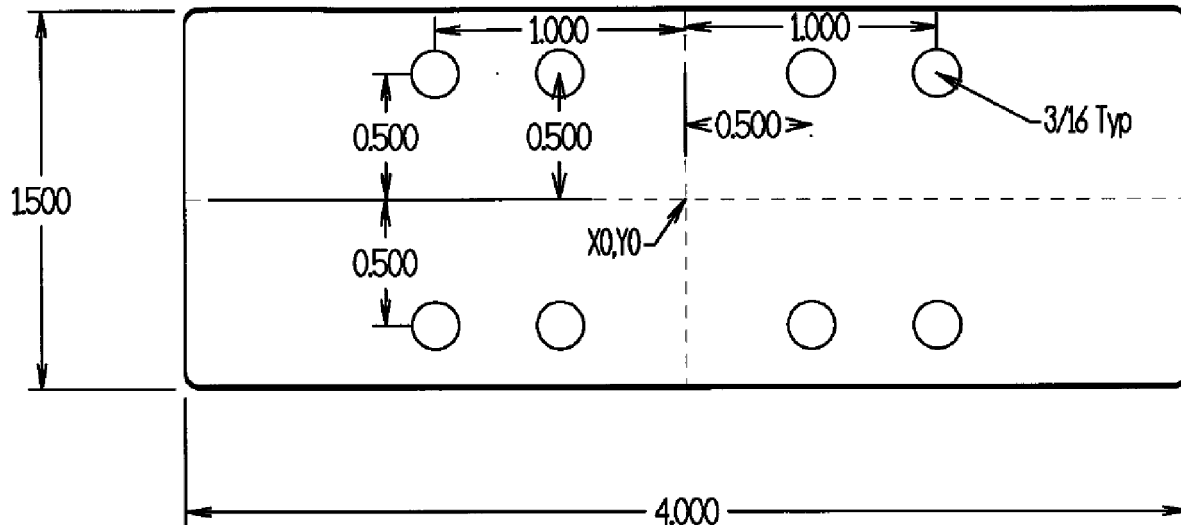
Part 102
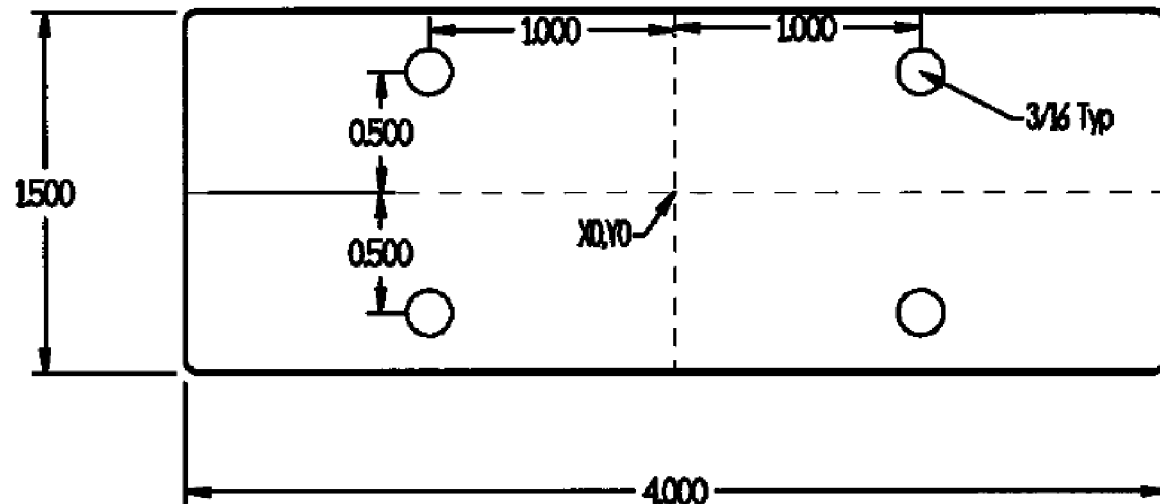
The only difference from Part 101 and Part 102 is that 4 interior holes have been subtracted from Part 102.

```
O101 (PART 101 AND 102)
G40 G49 G80 G90 (USE BLOCK DELETE FOR P/N 102)
T1 M06 (3/16 DIAM  DRILL)
G00 G90 G54 X-1.0 Y0.5 M03 S2400
G43 H01 Z1. M08
G83  Z-0.65 R0.1 F4. Q.2
/X-.5
/X.5
X1.0
Y-1.0
/X0.5
/X-.5
X-1.0
G00 G80 Z1. M09
M05
G00 G53 Z0
M30
%
```

Class Exercise: Block Delete

     Use block delete so the same program on page 28 may be used for cutting 1018 steel and aluminum
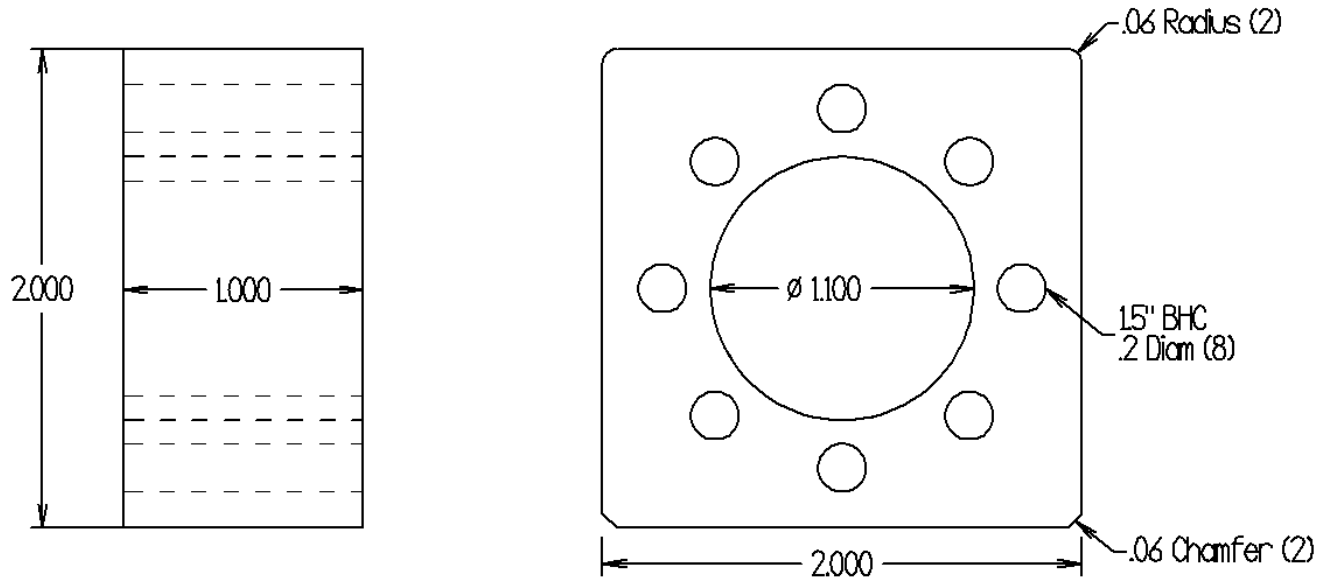
T1- 2" Face Mill (S4000, F50.)
     1018 Steel S500 ft/min, .0025/rev, 5 teeth

T2- 1" Carbide Insert Drill (S 675 Ft/Min) F .005/Rev
     1018 Steel S400 ft/min, .003"/rev

T3- Two Flute ½ Diameter Carbide End Mill (S800 Ft/Min, F .004 In/Rev)\
     1018 Steel S400 ft/min .004 in/rev
     Use arc on arc off to finish cut outside of part, Use G13 with G91 to finish ID Diameter to 1.100 inch step down 2 times.

T4- 3/8" Carbide Spot Drill 90 degree point, chamfer outside and inside bore, spot holes to leave .3 chamfer diameter.

T5- #7 High Speed Drill S 250 Ft/min F.004 In/Rev
     1018 S80 Ft/min F .005"/rev

T6- ¼-20 Tap
     1018 S30 ft/min

G68 Coordinate Rotation

Coordinate Rotation or Coordinate System Rotation allows a program that is described by orthogonal coordinates to be rotated along an axis. What this allows is a program that is described by axis perpendicular to each other to be machined at an angle. In the G17 mode the plane is described by X and Y coordinates in a typical top view drawing. With coordinate rotation a program may be simpler to create with these coordinates than refiguring the X and Y coordinates if the part is rotated when machined. See example below:
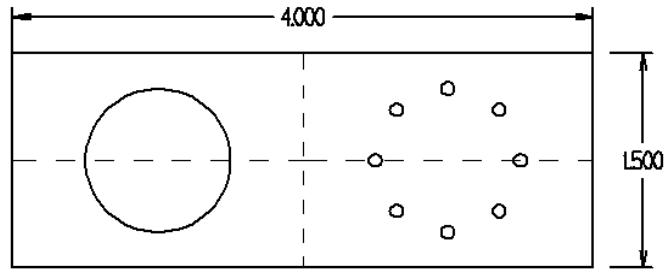
The above shows a top view drawing where center locations of pocket and bolt hole positions may be easy to calculate and write into a cnc program.
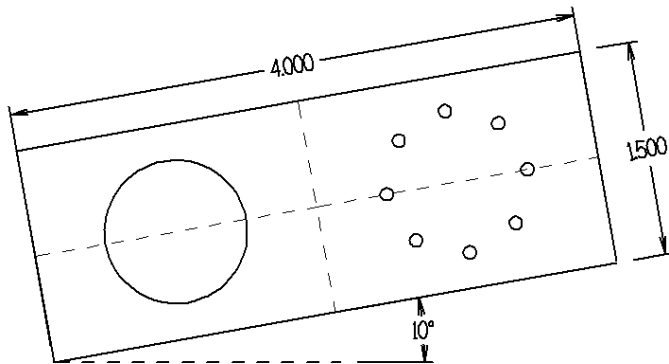
The first print was rotated 10 degrees counterclockwise around the lower left hand corner. As may be seen the X and Y coordinate calculations for the various features become complex and require the use of trigonometry functions.

Note: With Haas vertical machines Coordinate Rotation (G68) and Scaling (G51) are options that need to be purchased. However if the Haas vertical machine was purchased with a Renishaw Wireless Intuitive Probe System, Coordinate Rotation and Scaling are included.

G68 can be used in the G17, G18 or G19 planes.   G17, G18 or G19 must be called up in the program before the G68 command or the default G17 will be used.

The format for G68:   **G17 G68 Annn Bnnn Rnnn**

A and **B** describe the corresponding axis or center of rotation coordinates for the selected plane.   For G17 Xnnn and Ynnn would be used, G18 Xnnn and Znnn, G19 Ynnn Znnn. If no coordinates are called out on the G68 command line the axis of rotation will be at the current location of the machine.

R is the angle of rotation specified in degrees. Up to 3 decimals may be used. A **positive R** value indicates a **counterclockwise rotation.   Clockwise rotation** is directed by a **negative R** value.   If no R is given the machine defaults to the value given in **setting #72**.

Below shows the tool paths of the part on the previous page.   The left figure shows the part run normally. On the right the code G68 X-2. Y-.5 R10. has been inserted at the beginning of the program. The part zero is the center of the part. The coordinates for the lower left hand corner are X-2. Y-.5.   The tool path has been altered so that it is rotated 10° counter clockwise. The axis of rotation is the lower left hand corner of the part. Note the X and Y values in the G68 are not rotated.
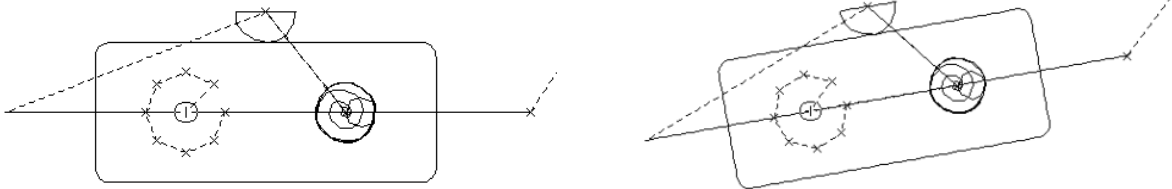
When using cutter compensation the **G41 or G42** commands should be turned on after the G68 rotation is called up and turned off with **G40 before** the **G68** is cancelled with a G69.

Applications

Rotation of part to fit work area on table

Below the part is longer than the work envelope of a Haas vertical machining center VF-3YT in X.
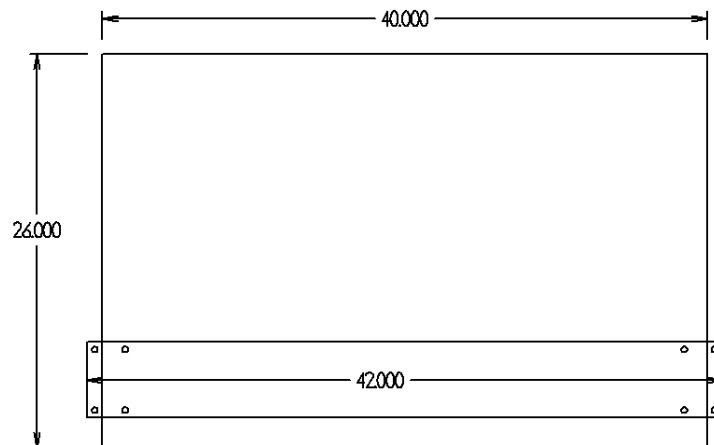
By rotating the part 30° **counterclockwise** the part now fits within the work envelope of the machine. The axis of rotation must be determined along with the appropriate angle of rotation to use **G68 +R.**
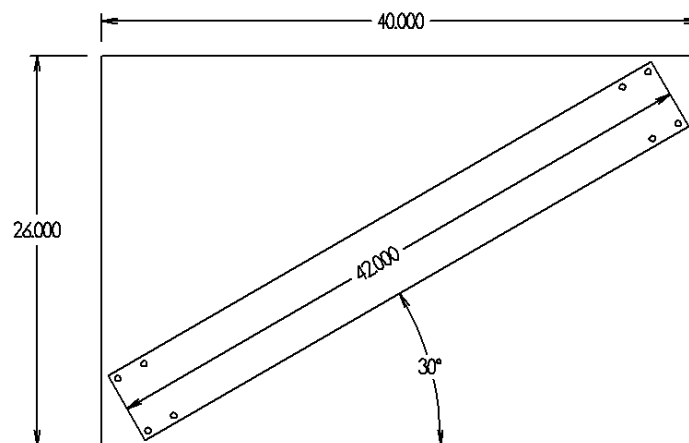
Below the Bolt Hole Circles are called out from the lower left hand corner of the part on the print. The part however will not fit on the table in that orientation.
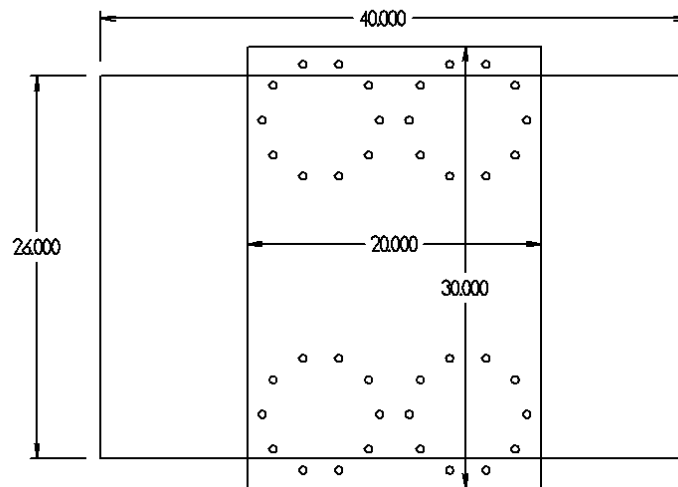
By rotating the part 90° clockwise the part will fit within the travel of the machine. In this example the part may be programmed with the part zero the lower left hand corner from the print. In the **clockwise** rotated state using **G68 R-90**. the upper left hand corner will then become the part zero reference point.
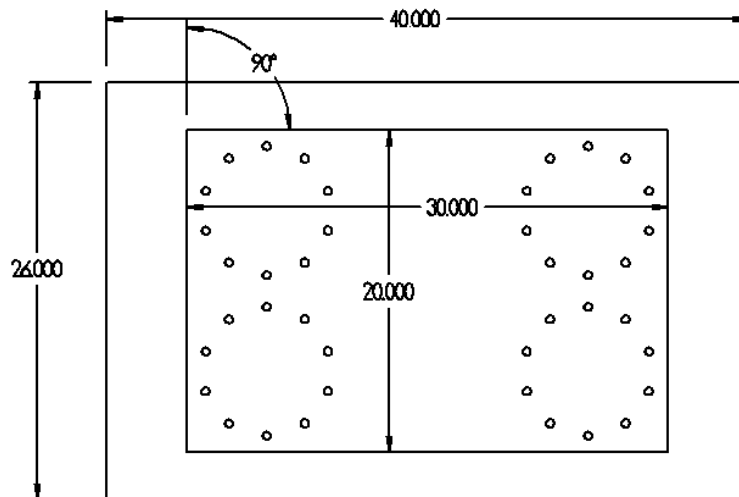
Incremental G68

Incremental G68 with sub routines may be used to program parts with repeating tool paths which rotate around some central point. To use incremental G68 **Setting #73 G68 INCREMENTAL ANGLE** must be turned to **ON** in Haas machines.

Below five ¾ in wide closed slots are located around the center of the part. A carbide 5/8 inch end mill is used to ramp down at 2° back and forth, finishes the bottom, and radiuses on and off to finish the .200 depth and the ¾ in width of the slots. The five slots are equally spaced 72° apart. The center of the part is designated the part zero.
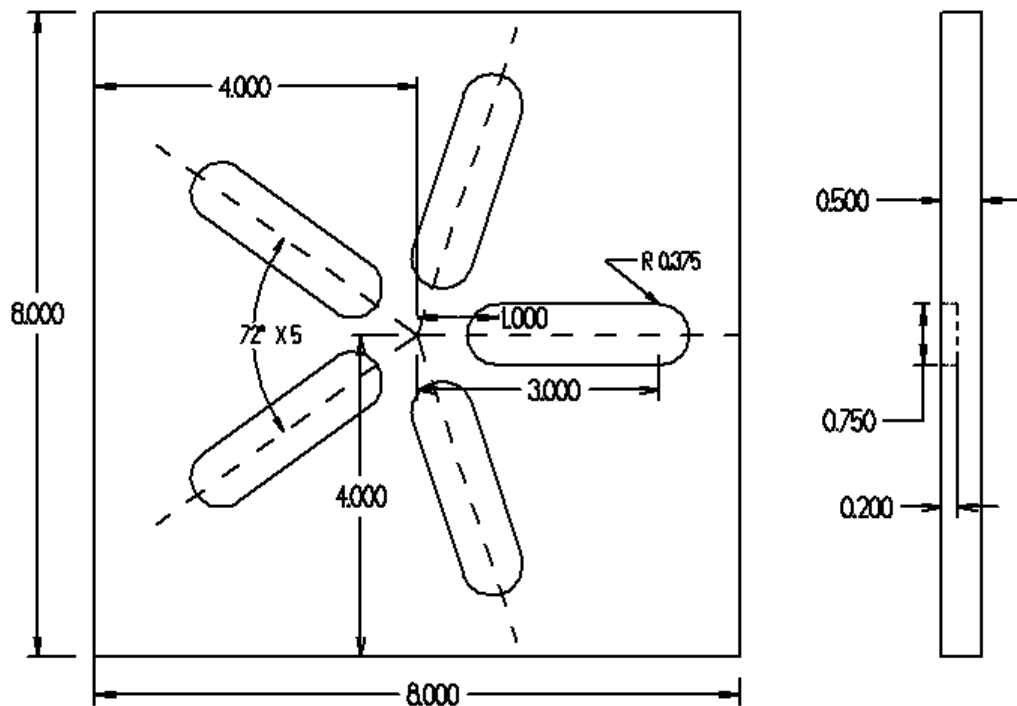
Key to the program is the incremental use of G68. Note the use of the following incremental command:

G91 G68 X0 Y0 R72. This command will incrementally rotate the tool path 72° counter clockwise to get the tool path of the closed slot in the upper right hand quadrant. By calling it up repeatedly using M98 Pxxxx **L4** all the slots may be simply programmed.

The following code may be used to program all five slots. First the tool is called up, a 5/8 carbide end mill in the main program. Then the sub program O03904 is called which describes the tool path of the slot at 3 o'clock. That slot will be machined ending with the tool 1 inch above the part. Then the sub program O3905 is call up 4 times using L4. Program O03905 incrementally rotates the coordinate 72° counter clockwise around the part zero (X0, Y0) then calls up the tool path program O03904. Then all the rest of the slots are created.

```
O03903 (CLOSED SLOT G68 MAIN PROG)
G28
T1 M06 (5/8 END MILL)
M98 P3904 (CALL TOOL PATH SUB)
M98 P3905 L4 (CALL ROTATION SUB  4 TIMES)
G91 G28 Z0 M05
G69 (CANCEL G68)
M30


O03904 (CLOSED SLOT TOOL PATH SUB)
N3 G90 G54 G00 X3. Y0 S9165 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z0 F80.
N6 X1. Z-0.07 F183.
N7 X3. Z-0.14
N8 X1. Z-0.2
N9 X3.
N10 G41 X2.65 Y-0.025 D01 F90.
N11 G03 X3. Y-0.375 R0.35
N12 Y0.375 R0.375
N13 G01 X1.
N14 G03 Y-0.375 R0.375
N15 G01 X3.
N16 G03 X3.35 Y-0.025 R0.35
N17 G40 G01 X3. Y0
N18 G00 Z1. M09
M99


O03905 (INC G68 SHIFT SUB)
G91 G68 X0 Y0 R72.
M98 P3904
M99
```

Graphic screen shot of above program
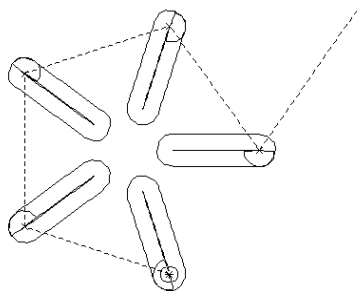
G51 Scaling

The Scaling Function is used when a program has already been created but needs to enlarged or shrunk to fill a particular need.

Examples of scaling function usage include:
- Similar parts programming with identical part geometry which are proportional to each other.
- Programming parts to allow for a shrinkage factor from heat treat or other processes.
- Fitting engraved characters or logos to a particular location and size.

The G51 scaling code is optional on Haas machines. Scaling is included with a Haas machine which has been purchased with probing.

G51 [X…] [Y…] [Z…] [P…]

> **X, Y, Z – Scaling Center** (If no XYZ center is called out on the G51 line the last position commanded is used as the scaling center.)

> **P – Scaling Factor** (three-place decimal factor from .001 to 8383.000) The P value multiplies all X,Y,Z, I,J,K or R values relative to the scaling center.

G50 – Cancels G51 scaling feature. Any axis scaled by previous G51 will be no longer in effect.

Particular attention must be paid to the **scaling center**. Below left is the tool path of the Productivity logo. To the right is the same tool path scaled 5 times with the lower left hand corner the scaling center.

.
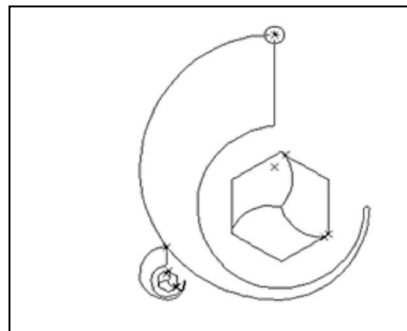
Below the   scaling center is the center of the logo.
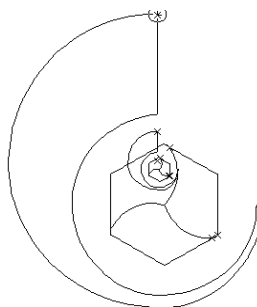
Datum Shift

In the print below we have a repeating bolt hold pattern in 4 different places. It would be nice to program the pattern once and repeat it three times just by shifting the center over. This is possible by using a datum shift. This can be performed by using local coordinate system G52. G52 is a coordinate system within a parent system (G54-G59). G52 is also called a child coordinate or global coordinate. What it does is any value in the G52 register will be added to any of the parent G54 to G59 or extra work offset values. What it does is temporarily transfer the part zero. In Haas milling machines G52 is refer as global work coordinate shift. It will be set to zero by a G92 command, when the power is turned on, or reset is pressed. After it is used it should be set back to zero with G52 X0 Y0 Z0.
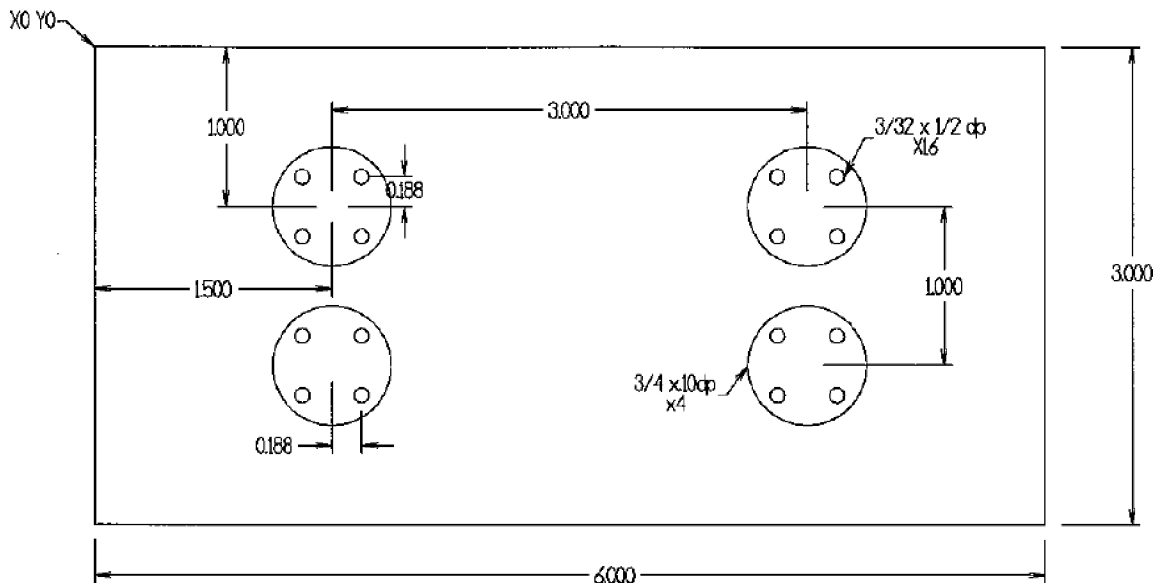
```
O1212
G00 G53 Z0
T3 M6 (3/32 DRILL)
G90G54 G00 X1.5 Y-1.0
M03 S1200
G43 Z1.0 H3 M08
G52 X1.5 Y-1.0
G98 G83 Z-.53 R0 Q.1 F10. L0
X.188 Y.188
X-.188
Y-.188
X.188
G80 Z1.0
G52  X4.5 Y-1.0
G00 X0 Y0
G98 G83 Z-.53 R0 Q.1 F10. L0
X.188 Y.188
X-.188
Y-.188
X.188
G80 Z1.0
G52 X4.5 Y-2.0
G00 X0 Y0
G98 G83 Z-.53 R0 Q.1 F10. L0
X.188 Y.188
X-.188
Y-.188
X.188
G80 Z1.0
G52 X1.5 Y-2.0
G00 X0 Y0
G98 G83 Z-.53 R0 Q.1 F10. L0
X.188 Y.188
X-.188
Y-.188
X.188
G80 Z1.0
G52 X0 Y0
G53 Z0 G49
M30
```

G10 Usage

Benefits of Setting Work Offsets, Tool Length, Cutter Compensation Values thru a Program

In small job shops the machine operator usually sets work, tool length and cutter compensation offset values during set up of different jobs. In high volume manufacturing where individual machines are dedicated to a family of parts or where a fixture is never removed from a machine data such as work, tool, and cutter compensation offsets can be saved and loaded into the machine thru the program using G10s. After the job is first set up data on work offsets, tool length offsets and cutter compensation is written into the program using G10s. The next time the job is set up work offsets, tool length offsets and D offsets are loaded into the machine thru the program.   When the G10 command is executed it writes over any value that is that particular register. This greatly reduces set up time when changing from one job to another. Normally the G10 command is in the first few lines of code at the beginning of the program.

G10 used in G90 mode writes over old offset value.
G10 used in G91 mode will add to the value already in the particular offset

Format:
G10 L** P** X** Y** Z**

> L2 -Work Offsets G52, G54-G59
> L20- Extra Work Offsets G110-G129, G154 P1 – G154 P99
> P- Selects Specific Offset

Work Offset	G10 Code	X	Y	Z
G52	G10 L2 P0 G90	X***	Y***	Z***
G54	G10 L2 P1 G90			
G55	G10 L2 P2 G90			
G56	G10 L2 P3 G90			
G57	G10 l2 P4 G90			
G58	G10L2 P5 G90			
G59	G10 L2 P6 G90			
G110	G10 L20 P1 G90			
G111	G10 L20 P2 G90			
G129	G10 L20 P20 G90			
G154 P1	G10 L20 P1 G90			
G154 P99	G10 L20 P99 G90			

Length Offset	G10 Code	Value
Tool #1	G10 L10 P1G90	R***
Tool #200	G10 L10 P200G90	R***

Diameter Offs	G10 Code	Value
Tool #1	G10 L12 P1 G90	R***
Tool #200	G10 L12 P200 G90	R***

Sub Routine Programs

Sub programs are used when some type of repetitive set of locations or cycles are used more than once. To keep the length of the program shorter the repetitive instructions or codes are separated out as a sub program or sub routine and called up when needed.   This not only keeps the program length shorter, it also is easier to write and verify the program. The shorter the program, the less likely errors will show up. They are used in repetitive motions, hole patterns, grooves and other special functions.

Subprogram call functions M97, M98, M99:

> **M97** – Local sub routine call, M97 PXXXX will jump to XXXX block number within the program that is being run.
>
> **M98** – Sub program call, M98 PXXXX will jump to program OXXXX.
>
> **M99** – Sub program end function. A M99 command returns to the main program in the next block of code after the M97 or M98 block.   M99 PXXXX will jump to a specific block number XXXX. M99 in the main program will cause the program to loop back to the beginning of the program without stopping.

Subprogram for repeating tool motion:

The most commonly used subprograms are for repeating tool motions or locations. A good example is the part found on page 26. The drilling part of the program can be separated out as a subroutine. See the example below.   G52 shift moves the X0 and Y0 over to the center of each bolt pattern and the subroutine drills the same pattern four times.

```
O1213 (G52 DATUM SHIFT WITH SUBPROGRAM)
G00 G53 Z0
T3 M6 (3/32 DRILL)
G90G54 G00 X1.5 Y-1.0
M03 S1200
G43 Z1.0 H3 M08
G52 X1.5 Y-1.0
M97 P10
G52  X4.5 Y-1.0
G00 X0 Y0
M97 P10
G52 X4.5 Y-2.0
G00 X0 Y0
M97 P10
G52 X1.5 Y-2.0
G00 X0 Y0
M97 P10
G52 X0 Y0
G53 Z0 G49
M30
```

```
N10 G98 G83 Z-.53 R0 Q.1 F10. L0 (DRILL SUB PROGRAM)
X.188 Y.188
X-.188
Y-.188
X.188
G80 Z1.0
M99
```

The same part can be programmed without work offset shifts if the drilling subroutine is programmed in G91 (Incremental Positioning). This creates an even shorter program.

```
O01214 (SUB G91)
G00 G53 Z0
T3 M06 (3/32 DRILL)
G90 G54 G00 X1.5 Y-1.
M03 S1200
G43 Z1. H03 M08
M97 P10
X4.5 Y-1.
M97 P10
X4.5 Y-2.
M97 P10
X1.5 Y-2.
M97 P10
G53 Z0 G49
M30
```

```
N10 G91 G98 G83 Z-0.53 R0 Q0.1 F10. L0
X0.188 Y0.188
X-0.375
Y-0.375
X0.375
G80 Z1.
G90
M99
```

Repeating Subprograms using L

If multiple depths of cuts need to be taken to create a feature, a subprogram can be developed using G91 which can be repeated multiple times using the L value.

M97 PXXX LXX will repeat subprogram NXXX LXX number of times.

The following example illustrates making a 13/16" diameter groove 1/16" wide .05" deep. If the material is a tough like D-2 tool steel a prudent programmer may take five individual cuts .010" deep.
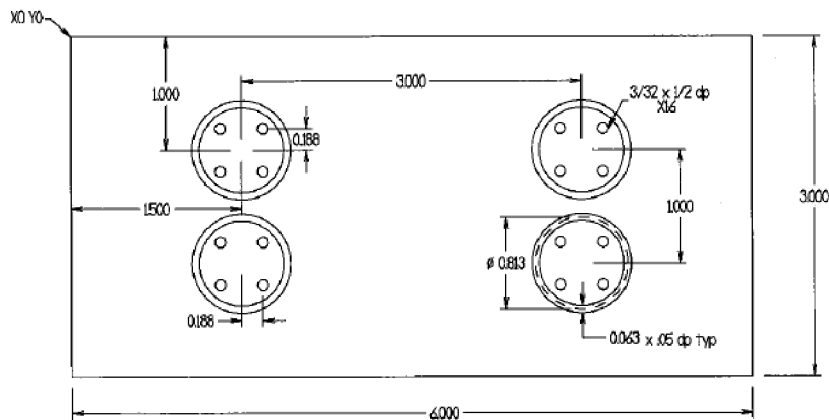
O01215 (SUB G91 WITH L)
G00 G53 Z0
T3 M06 (3/32 DRILL)
G90 G54 G00 X1.9062 Y-1.
M03 S3080
G43 Z1. H03 M08
Z.05
G01 Z0 F5.0
M97 P20 L5
G90 G00 Z.05
G0 X4.9062
G01 Z0 F5.0
M97 P20 L5
G90 G00 Z.05
Y-2.0
G01 Z0 F5.0
M97 P20 L5
G90 G00 Z.05
X1.9062
G01 Z0 F5.0
M97 P20 L5
G00 G53 Z0 G49
M30
N20 G91 G01 Z-.01 F.6
G03 I-.4062 F2.0
M99

Multi-Level Nesting Applications

In the examples shown so far the subprogram or subroutines do not call up another subprogram. This is referred to as one level nesting. If a subprogram calls up another subprogram it is called two level nesting. Below is program using two level nesting. The four pockets are milled with a .375 diameter 2 flute end mill. S.F.M.=200. FEED PER TOOTH=.004
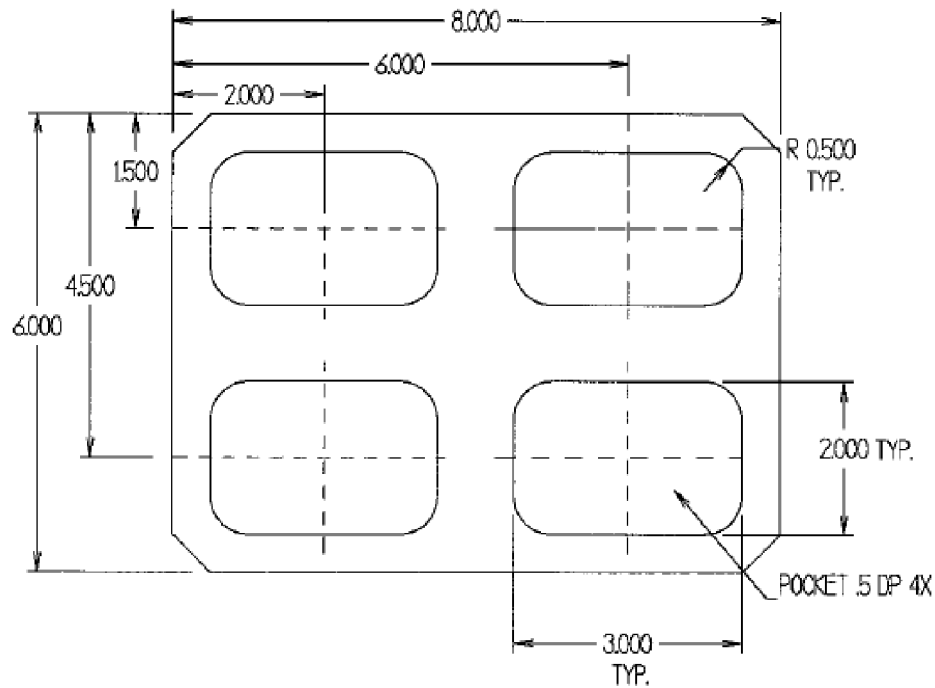

```
O01111 (MULTIPLE POCKET NESTING SUB ROUTINES)
G80 G90 G40 G49 G17
G00 G53 Z0
T1 M06 (.375 END MILL)
G00 G90 G54 X2. Y-1.5 M03 S2035
G43 H01 Z0.1 M08
G01 Z0 F50.
M98 P1000
G90 G00 Z0.1
X2. Y-4.5
M98 P1000
G90 G00 Z0.1
X6. Y-4.5
M98 P1000
G90 G00 Z0.1
X6. Y-1.5
M98 P1000
G90 G00 Z0.1 M09
G00 G53 Z0
G00 G53 Y0
M30
```

```
O01000 (POCKET SUB)
G150 R0.1 Z-0.5 I0.3 Q0.25 D01 F15. P2000 K0.01 G41
G91 G40 Y-0.1875
M99

O02000 (POCKET CONTOUR SUB)
G91 Y1.
X-1.
G03 X-0.5 Y-0.5 R0.5
G01 Y-1.
G03 X0.5 Y-0.5 R0.5
G01 X2.
G03 X0.5 Y0.5 R0.5
G01 Y1.
G03 X-0.5 Y0.5 R0.5
G01 X-1.
G90
M99
```

The main program takes the tool to the center of each pocket. Then it calls up the Pocket Sub program calling up G150 which is a pocket milling canned cycle. The G150 calls up another program which defines the pocket contour in G91 incremental positioning.

 Multiple 3, 4, 5 level nesting programs are possible. When nesting above level 2 it makes the program much more complex for machine operators to understand. When tools fail in the middle of the program the operator may not know where to restart the program.

Helical Milling

A helix is a curved movement around a cylinder with a simultaneous linear advance at a constant rate. Another definition is a spiral. It may also be described as any shape that resembles a screw. Below is a helix with four revolutions.
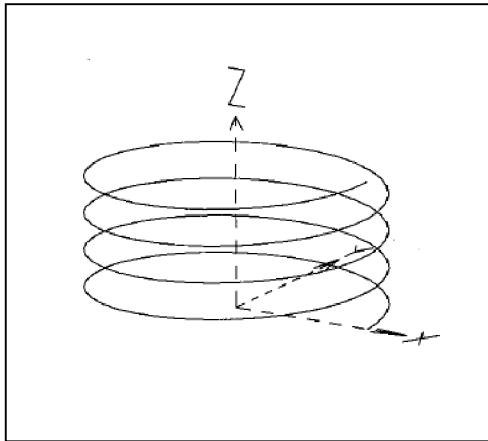
Isometric View
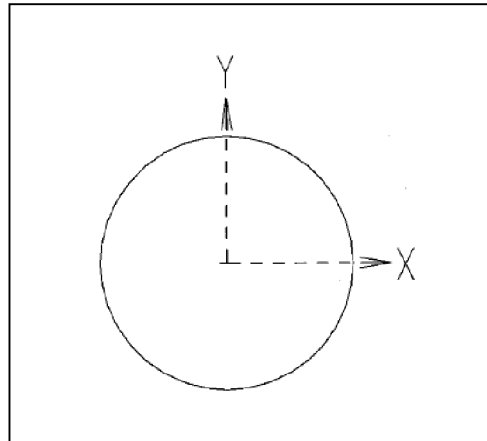
Top View
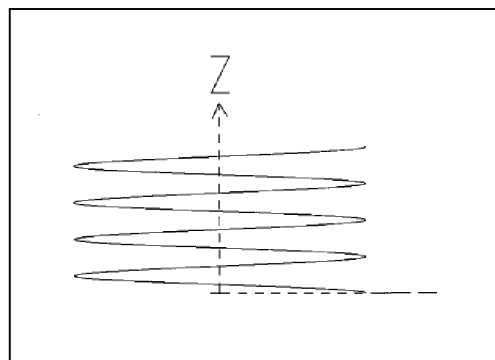
Front View

The program format for creating a helix : **G03 Xxxx Yxxx Zxxx Ibbb Jccc .**

X and Y are the start and finish location of the arc in the helix in **X** and **Y**. The **I** and **J** are the radial distance in X and Y respectively from the start position of the 360 degree arc to the center of the arc. The Z value is the linear motion which creates the helix. For any location the following code will create a one turn 1 inch diameter helix 1 inch high.

G91 G03 X0 Y0 I-.5 J0 Z1.0

The example is given with G17 active. G17 is the default code.   Most helical milling will be done in the G17 mode. Helixes may be created in G18 and G19. In G18 the circular motion is in the X and Z, the linear motion is in Y.   In G19 the linear motion is in X, the circular motion in Y and Z.

OD Thread Milling

In vertical machining centers tapping is the predominant method of creating threads. In some situations however it is difficult or impractical to use taps. In these instances thread milling may be the only way to create threads. A thread mill is a cutter formed with the pitch of the thread desired. Solid carbide cutters are fragile and expensive. Internal holes smaller than 3/8" may not possible or practical to create using solid carbide thread mills. For large threads, external threads, port threads, blind holes thread milling can be the most economical method. Climb cutting is generally used for better tool life.

Common uses of Thread Milling:
Large diameter threads: Any size thread may be created using thread milling.
Some machines lack the power at low RPM's to push large taps through.
Taps are not available in a particular size.
Tapping is difficult because of blind holes or hard material.
Left hand taps unavailable.
Longer tool life with thread mills.
Better surface finish with thread mills.
Easier or no thread deburring required.
One tool accepts inserts of different pitch sizes.
Cost reductions.

Thread Milling Tools (Examples taken from Carmex Precision Tools Ltd. X-Extreme Thread Cutting Catalog, Inch 2013, Carmex Precision Tools LLC, 2075 Hwy 175, Richfield, WI 53076)

Carmex Twin Insert Mill Thread Tool Holder

ID (Internal) Thread Milling

Threads are created using helical movements in machining centers. Using a standard G02 or G03 move with a Z move equal to the pitch of thread will create threads with corresponding diameters.   This creates one turn of a thread. With multiple teeth on a thread cutter more than one thread is cut.
The line of code below will create a one inch radius circle for a 20 pitch right hand thread.   This creates a 2 inch diameter thread with 20 threads per inch. Note that for a right handed thread the Z motion will be in the positive direction. For a left hand thread the Z helical motion will be in the –Z direction.

G02 I-1.0 Z.05 F5.

Tool Path X-Y Plane
Internal Thread Milling

Start Position and End Position

XO YO IS AT THE CENTER OF THE HOLE. ZO IS AT THE TOP OF THE PART
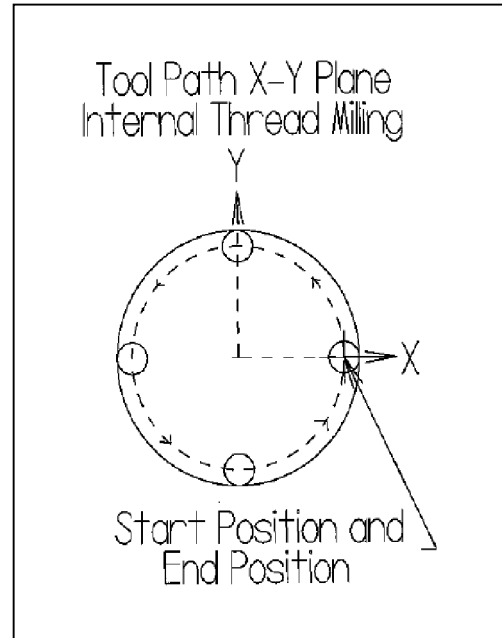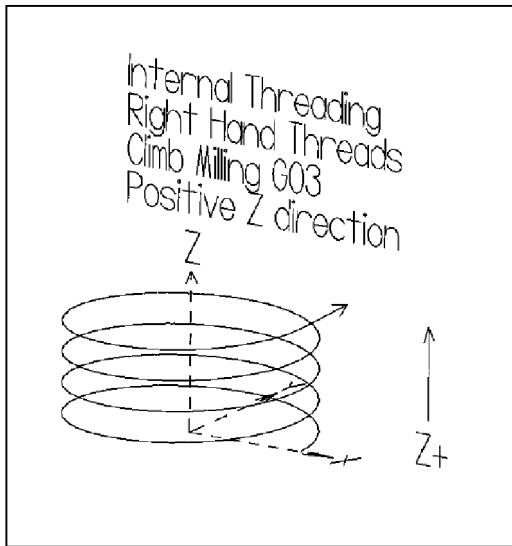
Lead in and Lead out motion
A common method involves 5 steps.
1) Activate cutter comp using 45 degree straight line movement ½ the radius of the finish diameter.
2) Arc movement into material, arc radius is ½ the radius of the finish diameter.
3) Circular motion counterclockwise 360 degrees cutting the thread. For right handed threads Z axis will move in the positive direction one thread pitch.
4) Arc movement out of the material similar to arc movement in.
5) Cancel cutter compensation with straight line movement back to center of the threaded hole.
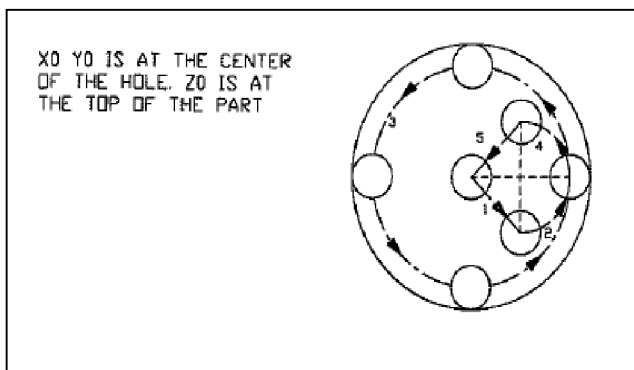
Creating the Code

Program G-Code for creating 2.0 inch diameter x 8 thread per inch through a hole .5 deep.
The hole is cut to the minor of the thread which is 1.849 in. Z0 is top face of part.   Hole is at coordinates of
X0, Y0. D2 is set to the diameter of the mill cutter
Cutter is a 2 flute with a diameter of .75
Surface Feet per Minute = 100
Feed per Tooth = .004

$$RPM = \frac{3.82 \times SFM}{Diameter} = \frac{3.82 \times 100}{.75} = \textbf{509}$$

Inch/min = RPM x Feed x # flutes = 509 x .004 x 2 = **4.07 in/min**

Due to the circular motion of the cutter the feed rate at the center of the tool will be different from edge of the tool.

Internal Threads: $F1 = \frac{F2 \times (Dw - Dc)}{Dw}$ External Threads $F1 = \frac{F2 \times (Dw + Dc)}{Dw}$

F1 = Programmed feed rate at center of tool
F2 = Actual feed rate desired at the cutting edge
Dw = Thread diameter
Dc = Cutter diameter

$$F1 = \frac{4.07 \times (2 - .75)}{2.0} = \textbf{2.5 in/min}$$

N1 T2 M6 (MILL THREAD)
N2 G00 G54 G90 X0 Y0 M03 S509   (Position to hole center in X and Y and turn on spindle)
N3 G43 H2 Z.1 M08 (Turn on tool length compensation, turn on coolant)
N4 G01 Z-.5156 F50. (Start depth plus 1/8 the pitch,
N5 G41 X.5 Y-.50 D2 (Activate cutter comp 45 degrees, ½ radius of finish diameter)
N6 G03 X1.0 Y0 I0 J.5 Z-.5 F2.5 (Arc on to finish diameter radius ½ the radius of finish diameter)
N7 G03 I-1.0 Z-.375 (Circular motion counter clockwise 360 degrees cutting thread, move Z positive one
                pitch length)
N8 G03 X.5 Y.5 I-.5 J0 Z.0156 (Arc off thread diameter ½ radius of the finish diameter, move up Z 1/8 x
                    pitch length)
N9 G01 G40 X0 Y0 F50. (Cancel cutter comp, move to center of thread)
N10 G53 Z0 M9
N11 M30

Thread Mill Exercise

Thread Mill a 1.5 Diameter x 10 TPI through hole .5 deep. Use a .5 diameter 4 flute thread mill using cutter comp. SFM = 400   Feed = .004/rev. Hole is cut to 1.379, the thread minor.

N1 T2 M6 (MILL THREAD)
N2 G00 G54 G90 X__ Y__ M03 S_____   (Position to hole center in X and Y and turn on spindle)
N3 G43 H2 Z.1 M08   (Turn on tool length compensation, turn on coolant)
N4 G01 Z-_____ F50.   (Start depth plus 1/8 the pitch,
N5 G41 X____ Y-___D2   (Activate cutter comp 45 degrees, ½ radius of finish diameter)
N6 G03 X___ Y___ I___ J   Z-___ F____   (Arc on to finish diameter radius ½ the radius of finish diameter)
N7 G03 I-___ Z-.___   (Circular motion counter clockwise 360 degrees cutting thread,
                                                        move Z positive one pitch length)
 N8 G03 X.__ Y.__ I-__ J_ Z_____   (Arc off thread diameter ½ radius of the finish diameter, move up Z
                                                        1/8 x pitch length)
N9 G01 G40 X0 Y0 F50.   (Cancel cutter comp, move to center of thread)

N10 G53 Z0 M9
N11 M30

Programming without tool radius in the D value

If the diameter of the thread mill is greater than the radius of the thread major the above method will not work as the G41 move from uncompensated to compensated will be less than the radius of the thread mill. In this case the thread mill tool path is programmed in G91 (incremental) and the D value is set to 0. See below:
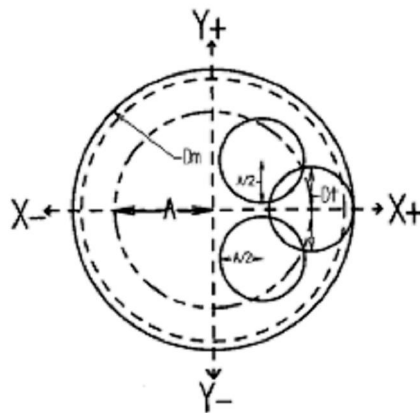
$$A=\frac{Dm-Dt}{2}$$ A=Radius of Tool Path
                    Dm=Major Diameter of Thread
                    Dt=Diameter of Tool

General Program

```
G90 G00 G54 G43 H1X0 Y0 Z10 S---
G00 Z- ( TO THREAD DEPTH )
G01 G91 G41 D1 X(A/2) Y-(A/2) Z0 F---
G03 X(A/2) Y(A/2) R(A/2) Z(1/8 PITCH)
G03 X0 Y0 I-(A) J0 Z(PITCH)
G03 X-(A/2) Y(A/2) R(A/2) Z(1/8 PITCH)
G01 G40 X-(A/2) Y-(A/2) Z0
G90 X0 Y0 Z0
```

Internal Thread
```
EXAMPLE : 11/4-12UN (Thread depth .71)
TOOLHOLDER : SR0790 H21 (Cutting Dia. .79)
INSERT: 21 I 12 UN
A = (1.25 - .79)/2 = .23
G90 G00 G54 G43 H1X0 Y0 Z0.39 S2800
G00 Z-0.71
G01 G91 G41X0.1150 Y-0.1150 Z0 F3.35 D1
G03 X0.1150
Y0.1150 R0.1150 Z0.0104
G03 X0 Y0 I-0.23 J0 Z0.0833
G03 X-0.1150 Y0.1150 R0.1150 Z0.0104
G01 G40 X-0.1150 Y-0.1150 Z0
G90 G0 X0 Y0 Z0
```

The above example is taken from "Carmex Precision Tools Ltd 2012 General Catalog".   Instead of taking the time to create the code oneself suppliers of thread cutting tools will provide free software for the asking or may be down loaded from their websites. They provide information such as optimum feeds and speeds. They also provide information on optimum size cutters per particular thread sizes.

External Threads

For climb cutting external right hand threads a clockwise helical move G02 is made in the negative Z direction. See figure below. In this example the tool makes a tangential arc on to the minor diameter of the thread. Then is makes a clockwise move G02 to cut the thread. For a left hand thread a positive Z helical movement would be required.
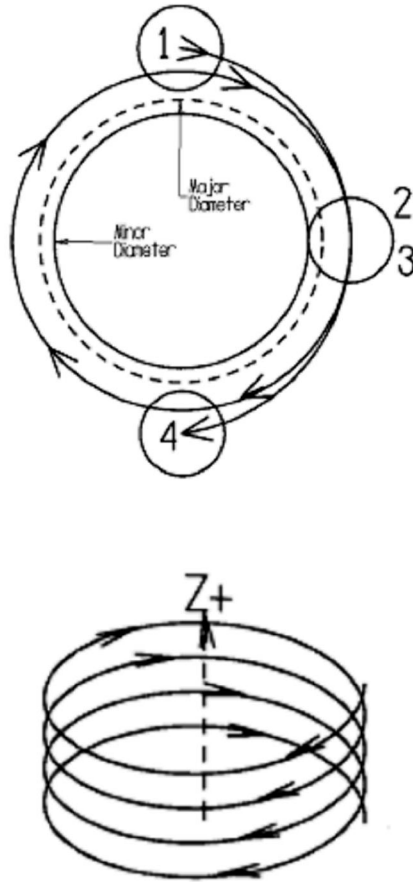
As noted on the previous page various thread mill tool suppliers software will create code for external and internal threading.   The following incremental thread program for an external UN 1-12 was created from software downloaded from Advent Tool and Mfg. Inc. website: **www.advent-threadmill.com**
http://www.advent-threadmill.com/downloads.html

```
%
N5 O1234
N10 G90 G0 G17 G40 D0 G54 G20 G80 G94
N15 ( CREATED BY ADVENT THREAD MILLING APPLICATION )
N20 ( THIS PROGRAM IS PRODUCED WITH NOMINAL NUMBERS. )
N25 ( YOU MUST ADJUST WITH YOUR OFFSET FOR YOUR PERFECT SIZE! )
N30 ( TOOL CENTER PROGRAM SET TOOL OFFSET D = 0)
N35 ( UN 1-12 RH OUTER THREAD IN 420 )
N40 ( TOOL=01-TA-01-F3-9 w/ ATM-38B 12)
N45 ( CUTTING SPEED=240, CHIPLOAD=0.0012 )
N50 ( FEED AT CUTTING EDGE=11, RPM=3055 )
N55 ( AT D0.9008 TOOL CENTER FEED = F14.7 )
N60 ( INCREMENTAL PROGRAM-CLIMB MILLING CODE FOR Fanuc 0M )
N65 T01
N70 M6
N75 G43 Z0.1000 H1
N80 S3055 M3
N85 G91
N90 Z0.0000
N100 M8
N105 G1 X0.0073 Y0.6802 Z0.0000
N110 X0.0000 Y0.0000 Z-1.1958
N115 G41 D21 X-0.0002 Y-0.0200 Z0.0000 F11.0
N120 G2 X0.5933 Y-0.6602 Z-0.0208 I-0.0706 J-0.6602 F13.0
N125 X0.0000 Y0.0000 Z-0.0833 I-0.6004 J-0.0000 F13.2
N130 X-0.5933 Y-0.6602 Z-0.0208 I-0.6639 J-0.0000 F13.0
N135 G1 G40 X0.0021 Y-0.0199 Z0.0000 F40.0
N140 G0 G90 Z0.1000 M5
N145 M9
N150 M30
%
```

Helical Ramping

Helical interpolation can be used for drilling or plunge cutting into solid materials. Normally it is necessary to pre drill a hole undersize and then take the hole to size with an end mill. If an end mill is center cut ground it may also be plunged into solid material however it is not very fast and tool life may be limited. In the above situations all the cutting is done on the end of the tool. Using helical ramping the cutting is done on the cutter sides not the bottom. With helical ramping a series of circular motions moving along with incremental pitch Z motions remove the material. Helical interpolation can be used to drill holes of various sizes with only one tool. It is good for creating a hole with interrupted cuts. It also requires less power to create the hole with respect to drilling.

Generally speaking the cutter used should be a shoulder type cutter (90 degrees). Maximum diameter to cut is twice the cutter diameter. Smallest recommended cut in the range of 1.1 to 1.3 times the diameter. The depth of cut is referred to as the pitch. This would be the z cut depth per revolution. In inch the maximum range of recommended cuts are .047" to .300" depending on the cutter geometry. Consult your tool supplier representatives. Most CAM software programs have capability to created hole using helical ramping.

Below are illustrations of tool paths using Helical Milling in Drill applications, creating a hole using helical ramping.
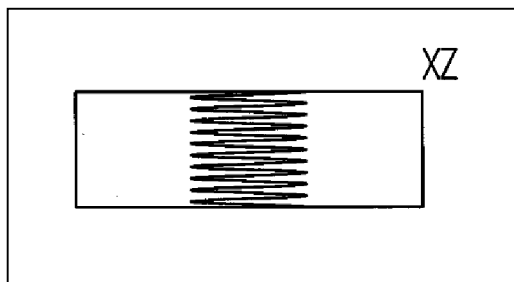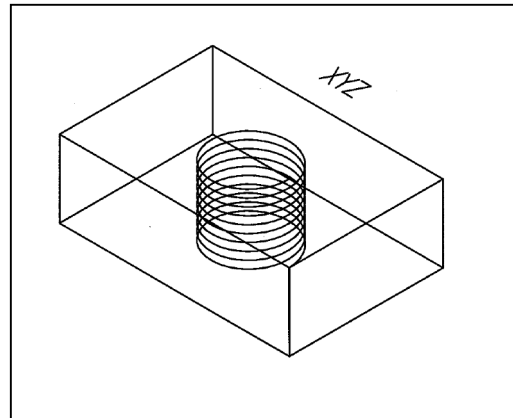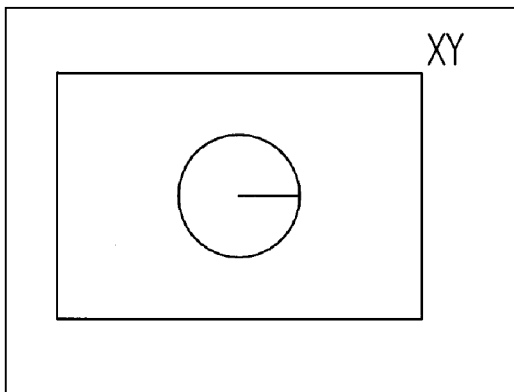
```
%
O00011 (HELICAL RAMPING)
G00 G17 G40 G49 G80 G90 G98 (HELICAL RAMPING TO DRILL 1" HOLE)
G00 G53 Z0
T17 M06 (5/8 ENDMILL)
G90 G54 G00 X0 Y0
M03 S1200
G43 H17 Z0.1 M08
G01 Z0.1 F50.
G41 X0.5 D17 F11.5
G91 G03 I-0.5 Z-0.1 L12
G90 G01 G40 X0
G00 Z1. M09
G91 G28 Z0 M05
M30
%
```

Ramping techniques used similar to helical ramping may be applied to many shapes like ovals, squares, rectangles, diamonds and many different shapes. Ramping is also used when cutting features in hard materials such as D-2 tool steels.

4ᵗʰ Axis Machining (Milling)

Rotary devices are used on vertical as well as horizontal milling machine centers to present several sides of the work piece to the spindle. This reduces the number of set ups and programs needed to manufacture a part. Reducing the number of set ups also reduces machine down time increasing the overall the number of parts that may be produced over a certain amount of time.

 A second advantage of using a rotary device is that the overall accuracy from one surface to next is improved. With multiple set ups it is difficult to locate the part perfectly. As the part is not removed using a rotary device overall accuracy increases.

There are two types of rotary devices: **indexers** and **rotary tables**. An indexer will rotate a part rapidly a prescribed number of degrees. Then machining operations are done after a brake is applied. Rotary devices are more flexible in that they not only will index to a particular angle but also may be rotated at a particular angular feed rate. This allows for machining along the angular axis of a part. When integrated with the host machine controls simultaneous multi axis machining becomes possible.

Indexers

Below is a Haas HA5C indexer. The resolution is .001°, speed .001° to 410° per second. It uses standard 5C collets with a manual hand or pneumatic closer. Small chucks may also be attached. Normally it is aligned along the X axis with the handle to the right. I may be also aligned with either the Y or Z axis. Common uses include cutting wrench flats, keyways and pin holes on shafts. Convention has that if extra axis is added A would correspond to rotation along the X axis, B along Y axis and C along the Z axis. In practice if only one rotary axis is added it will be the A axis however it is aligned be it the X, Y or Z axis.

Illustration taken from Haas 96-0315r rotary.pdf - Haas Rotary/Tailstock Operators Manual December 2012

HA5C Mounting
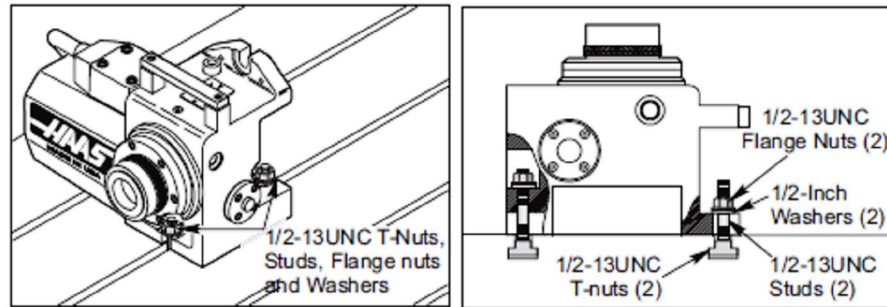
Above shows the HA5C mounted along the X-axis of a vertical machining center.

Haas indexers are set up two different ways: Semi-Fourth and Full Fourth Axis Operation. In the semi-fourth mode a separate servo controller with a completely separate program controls the rotation of the work piece. The host machine sends out a M21 command, the indexer rotates the amount indicated in its internal program and then sends a finish signal back to the host machine. The host machine then performs a machining operation and then sends another M21 command to the servo controller. The servo controller then rotates the head the number of degrees indicated in its 2nd line of program and sends back a finish signal. Then machining is performed by the host machine at that particular angular location. In the "semi-fourth axis" mode the rotary device cannot do simultaneous interpolations with X, Y or Z axis. Only when the rotary unit is directly interfaced with the host machine may full fourth axis operation be performed. See illustration below.
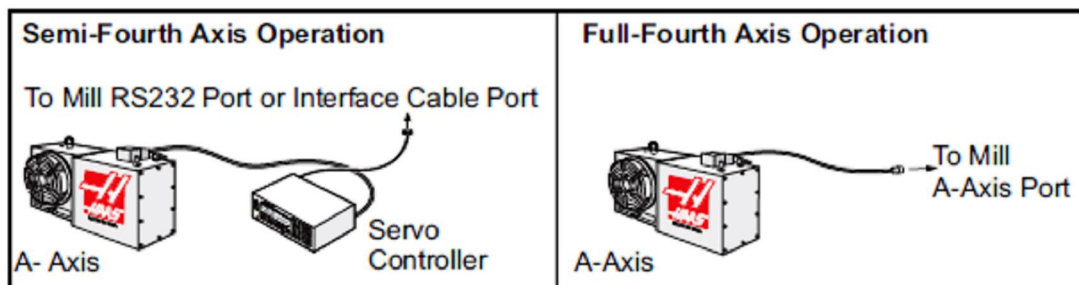
Illustrations taken from Haas 96-0315r rotary pdf - Haas Rotary/Tailstock Operators Manual December 2012

Semi-Fourth Axis Operation

To drill four holes 90° apart in illustration on the next page first the CNC mill must be positioned to a start drilling position on the part. With respect to the HC5C indexer the part is sticking out on the left.   G54 has been selected with X0 being the end of the part and Y0 the centerline of the part and Z0 the top of the part.

The servo controller program consists of 5 different pieces of information: STEP#, STEP SIZE, FEED RATE, LOOP COUNT, and G CODE. Refer to the manual on how to key in the program.

STEP SIZE is in degrees out to 3 decimals,
FEED RATE is in degrees per second
LOOP COUNT is the number of time Step line is to be repeated.
G CODE can be several different types. Probably the most commonly used are G90- Absolute Mode, G91- Incremental mode and G88- Return to home position G99- end of program, return to beginning.
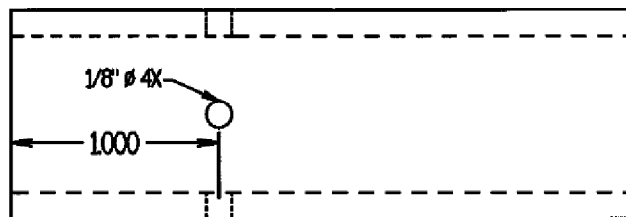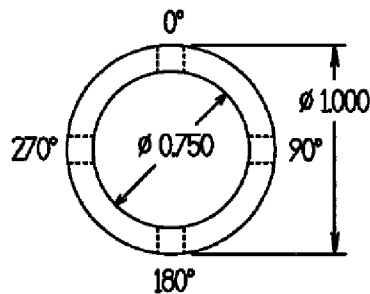
In the main program an optional user M code interface with M-fin signal is used to call up the indexer program.   Commonly used is M21.

O1 (DRILL 4 X 1/8" 90° APART)
G0 G53 Z0
T1 M6
G0 G54 G90 X1.0 Y0 M3 S1833
G0 G43 Z1.0 M8
G81 Z-.185 R.1 F5.5
M21 (CALLS UP INDEXER PROGRAM)
G80
M21 (INDEXER TO 0)
M21 (REWIND INDEX PROGRAM)
G00 Z1.0 M09
G00 G49 G53 Z0
M30

HAAS SERVO CONTROLLER PROGRAM

STEP#	STEP SIZE	FEED RATE	LOOP COUNT	G CODE
01	[90.000]	080.000	3	[G91]
02	0	080.000	1	[G90]
03	0	080.000	1	[G99]

With full fourth axis operation rotation of the indexer is called out in the main program.
The program to drill the same part with fully integrated fourth axis:

```
O1 (DRILL 4 X 1/8" 90⁰ APART)
G0 G53 Z0
T1 M6
G0 G54 G90 X1.0 Y0 M3 S1833
G0 G43 Z1.0 M8
A0
G81 Z-.185 R.1 F5.5
A90
A180
A270
G00 G80 Z1.0 M09
A0 (INDEXER TO 0)
G00 G49 G53 Z0
M30
```
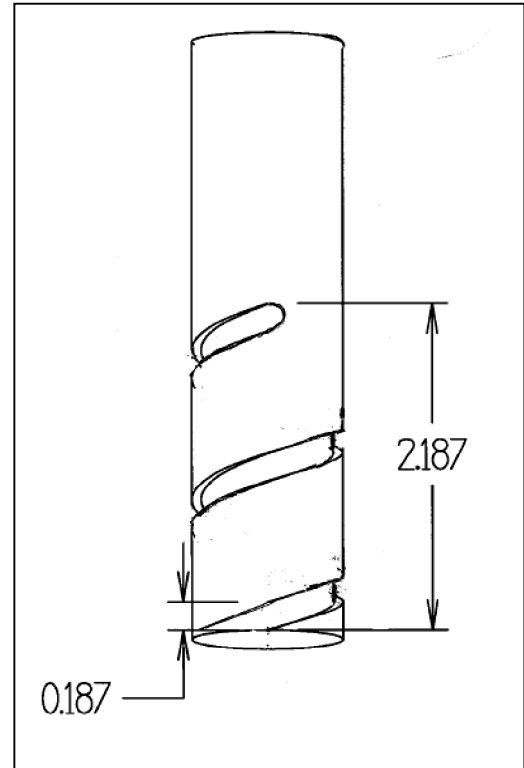

Full 4ᵗʰ Axis Rotation

Spiral grooving is possible with full 4ᵗʰ axis rotation. To make the part to the right simultaneous angular motion of rotary unit must be coordinated with a mill axis. At right a 3/16 wide groove .06" deep is spirally cut on 1"diameter round stock. The pitch of the spiral cut is 1".

The following calculations if using Haas Servo Controller:

Calculation of feed rate in degrees/second:
1) Calculate <u>radial move of cut in inches</u>
   Calculate degrees movement= number of revolutions x 360
   With X0 the end of part, the start position in X-.093
   End position in X=2.187-(.187/2)=X2.0935
   Total distance travelled in **X**=.093+2.0935= **2.187 inches**

   Degrees travelled: $\dfrac{2.187 \times 360 \text{ degrees}}{\text{Pitch}(1)}$ = **787.32 degrees**
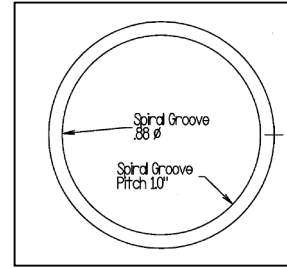
   Total distance cutter travelled over material when not moving in x-axis.

   Distance= $\dfrac{\pi \times \text{diameter} \times \text{Number of degrees}}{360 \text{ degrees}}$ = $\dfrac{3.1416 \times .88 \times 787.14}{360}$ = **6.045 inches**
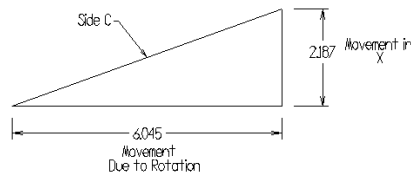
$$C^2 = A^2 + B^2$$
_____
Resultant move = $\sqrt{(2.187)^2+(6.045)^2}$ = **6.428 in. Total Movement**

2) Calculate RPM and Feed Rated in Inches/min:
   $$\frac{\text{Rev}}{\text{Min}} = \frac{\textbf{3.82 x S}}{\textbf{Diameter}}$$

   RPM=3.82x325/.125=**9932 rev/min**

   Calculate IPM= RPM x Feed /rev x #Teeth= 9932x.0006 x 4 = 23.8 in/min

3) Calculate time to complete movement = Distance/Inch per Min = 6.428/23.8=**.270min**
   = **16.2sec**

4) Calculate Feed Rate in Degrees per second = degrees travelled/time = 787.14/60x.270min
   = **48.58 degrees/second**

Computation of feed rate in in/min for the mill to travel the X distance in 16.2 seconds.

   Feed Rate (in/min) = $\dfrac{\text{length of travel x 60 seconds}}{\text{\# seconds rotation}}$ = $\dfrac{6.045 \times 60}{16.2}$ = **22.38 in/min**

With the servo controller the indexer needs to step **787.32 degrees** at a feed rate of **48.58 degrees/second.**

_____

The G94 command is used with the servo controller to control simultaneous angular motion of the indexer with axis motion of the mill.   G94 in the control program is followed by degrees of rotation and the feed rate in degrees per second in the next step of the program.

STEP	STEP SIZE	FEED RATE	LOOP COUNT	G CODE
01	0	270.000	1	**[94]**
02	**[-787.32] [48.35]**	**1**	1	**[91]**
03	0	270.000	1	**[88]**
04	0	270.000	1	**[99]**

G94 pulses the MFIN relay and allows the CNC to proceed.

The program in the CNC mill:

N1 G54 G90 G00 X-.0937 Y0 (rapid to start position in X and Y)
N2 M03 S9932
N3 G43 H1 Z.100 M08
N4 G01 Z-.06 F22.38 (feed to depth)
N5 G00 G91 (rapid in incremental mode)
N6 G01 F10. Z-.06 (feed down to Z depth)
N7 M21 (to start indexing program above at Step 1)
N8 X2.0935 F22.38 (index head and mill move at same time here)
N9 G00 Z1.0 (rapid back in Z axis)
N10 M21 (return indexer home at Step 3)
N11 G0 G53 Z0
N11 M30

In Full Fourth Integration with the Haas Mill the indexer is hooked into the fourth axis port. In the Haas mill the feed rate in the program is always in in/min. or mm/min. To obtain an angular feed rate in the 4th axis setting **#34** is used. The diameter in setting **#34** is used to determine the angular feed rate by the Haas control to correspond to the feed rate given in inch/min.

The program in Haas mill: (setting #34 = .88 inch)

T1M6
 N1 G54 G90 G00 X-.0937 Y0 (rapid to start position in X and Y)
N2 M03 S9932
N3 G43 H1 Z.100 M08
N4 G01 Z-.06 F23.8 (feed to depth)
N5 G01 F23.8 Z-.06 (feed down to Z depth)
N6 X2.0935 F23.8 A-787.32 (index head and mill move at same time here)
N7 G00 Z1.0 (rapid back in Z axis)
N8 G0 G53 Z0
N9 M30

Note the direction of the rotation of the head is in the **negative** direction.

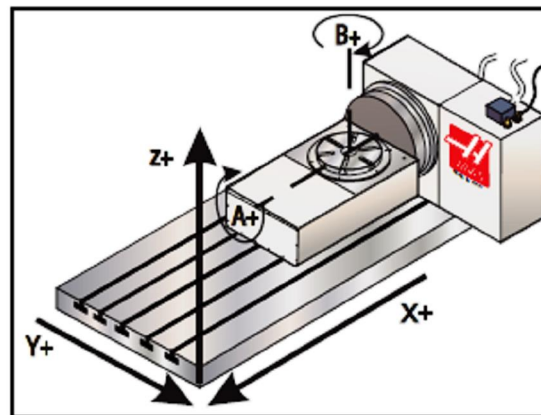**Extra Axis Coordinate System**

The illustration below at first glance gives confusing information. By convention an A axis rotates around the X-axis. Using the right hand rule when the thumb is pointed in the positive X direction a positive rotation will be in the direction of the fingers are pointed. This is indicated in the illustration below right. Another way to look at it is, if you are facing the rotary head, a positive rotation is clockwise. Note bottom left figure gives the direction of the positive direction of A axis work coordinates. They go counter clockwise. That is 90° is at 9 o'clock position, 180° is at 6 o'clock position, 270° is at 3 o'clock position. It is similar to where a positive movement of the table in the X or Y axis is opposite the positive direction of the work coordinates.

For the B axis to be lined up with the Y axis below the A axis would have to be rotated to the +90 degree position on the trunion table below. Then the right hand rule would apply as above. This will only apply if the rotary unit is positioned to the right.
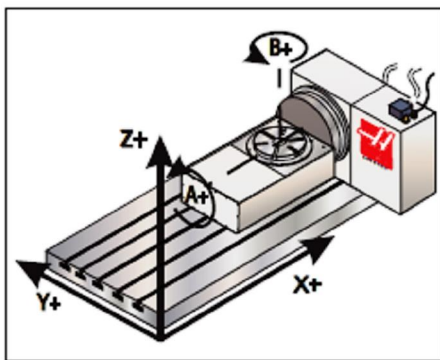
Work coordinates (Positive direction).   Table movement (Positive command).

The following illustrations would be from the left side view point of a Haas HRT160 Rotary Table with a Tooling Block System attached below.
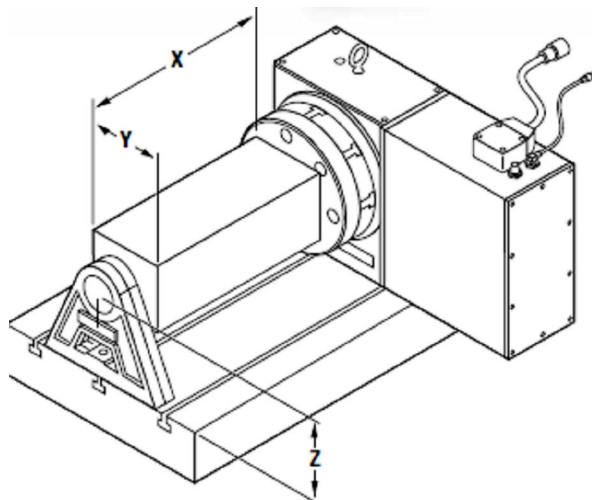
The above illustrations are taken from Haas Rotary/Tailstock Operators Manual, December 2012.

The graphics below shows an absolute G90 clockwise positive rotation to an A90 position from A0 position viewed from the left side.
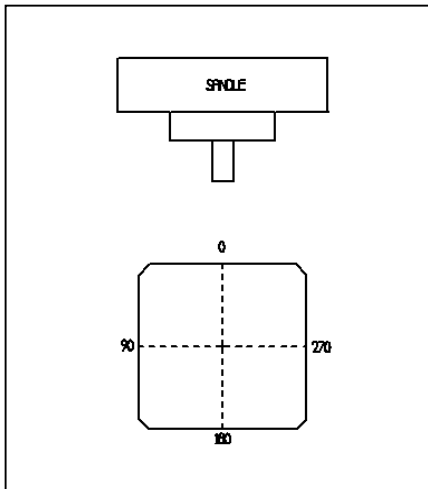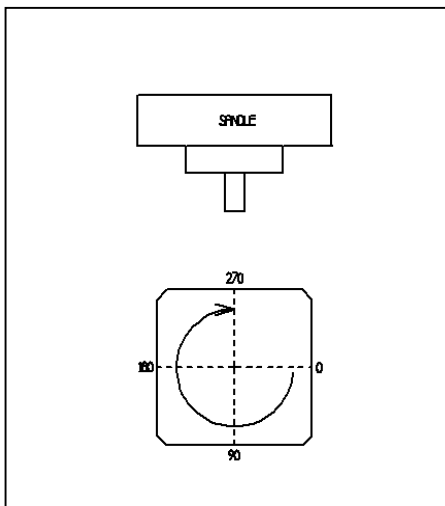

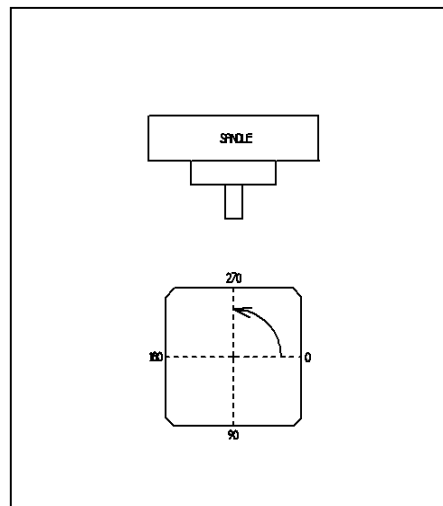Left side view of rotary table at A0 (home position)　　　　G90 G00 A90 (Absolute A90 Move)
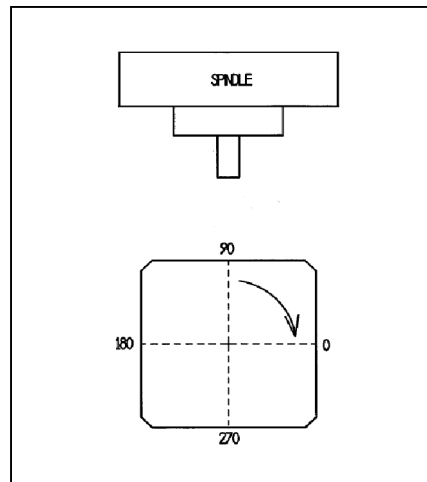
Below left shows an absolute move from A0 to A270. The faster way would be to put the machine in G91 and incrementally move G91 A-90 as shown bottom right. It still presents the A270 side to the spindle.

G90 move to A270 from A0　　　　　　　　　　G91 move of A-90 from A0

If a G91 is used to index the rotary unit a G90 must follow to get the machine back in absolute to use the correct work offsets for machining. Sometimes the incremental mode is the best for simple indexing. It gives control over which way the indexing occurs.

The Haas limits the A axis from A-8380. to A8380. The resolution is to $.001^o$. Rapid rotations run from 50° per second on large rotary tables to 725° per second on smaller units.

One of the disadvantages with progressive incremental moves in the rotary axes is the accumulation of large A degree values. If four tools were used on each of the four sides and only incrementally one way large A values could accumulate: 4 x 4 x 90 = 1440°. If the rotary tables is commanded to go home to A0 if may take a considerable amount of time for the rotary table to unwind itself. Large A values may also accumulate when long spiral grooves are machined.

To not require extensive time devoted to unwinding the A axis Haas has developed the **Quick Rotary G28** feature. If the table is at an A370^o the table will just rotate back A-10° with a **G28** or a **G91 G28 A0** command. It won't rotate back A-370°.

To use this feature **Setting #108** must be turned **ON.** Also, **Parameters 43:10 CIRC.WRAP**. and **151:10 CIR. WRAP. must be set to 1**.

Central Zero Program Method
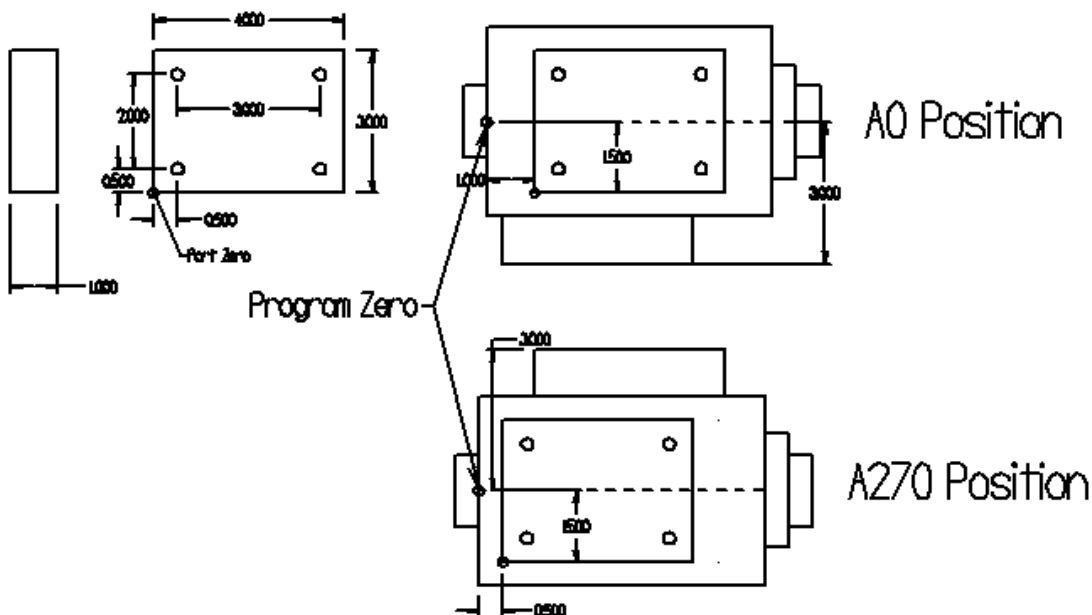
Above shows a 4" x 4" cubical fixture (similar to the one shown on page 49) holding 3" x 4" x 1" blocks where four holes are to be drilled. The rotary head would be to the right. The upper left print gives dimensions of the part to be drilled. The program zero point in X is the left side of the fixture. The Y and Z zeros are the center of rotation which lies along the Y axis. Using this method the programmer needs to add the difference (Δ) in each X, Y, Z coordinate from program zero. Then, add the hole locations as shown in the print in the upper left.

For A0 position ΔX=1.0 ΔY-1.5 ΔZ=3.0 Lower hole location on top of part X1.5 Y-1.0 Z3.0
For A270 position ΔX=.5 ΔY=-1.5 ΔZ= 3.0. The lower hole location on top of part X.5 Y-1.0 Z3.0
Although it may make it easier for the operator to set up with only one work offset the programming becomes much more difficult. If the fixture is not made accurately further corrections need to be made to the program. The most commonly used method is to have a separate fixture offset for each work piece.

Multiple Fixture Offset Method
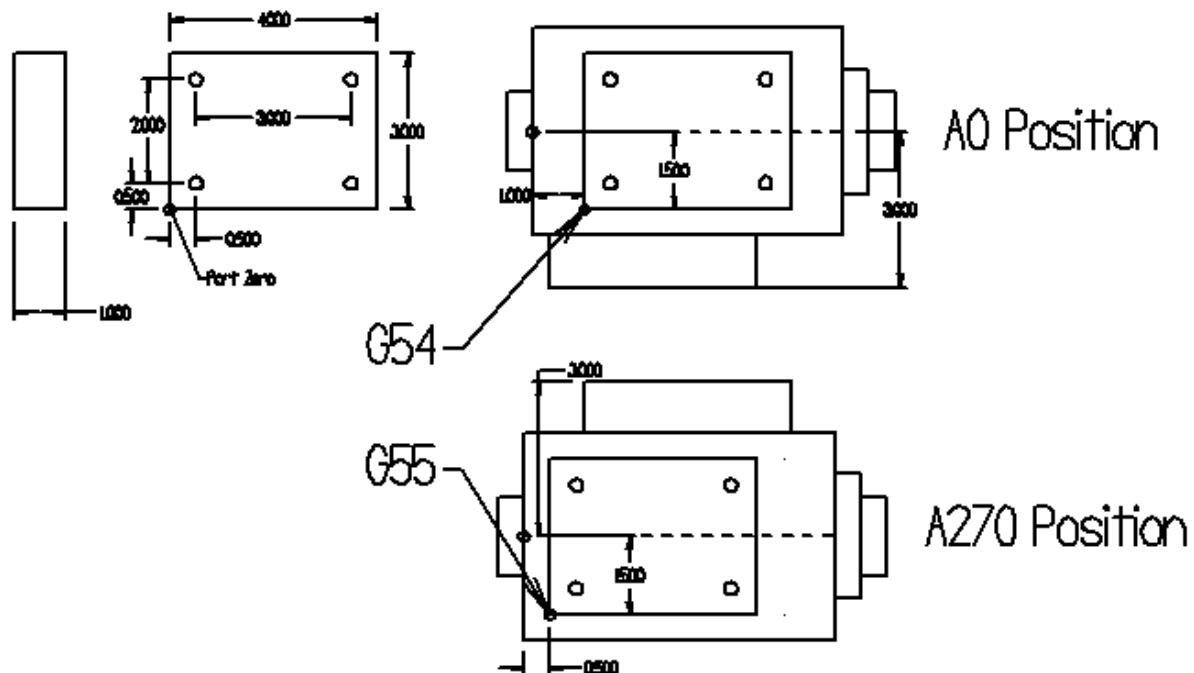
Above shows locations where work offsets G54 and G55 are used for each work piece placed on different sides of a rotary fixture. The locations reflect the datum point where all the dimensions come off the print. The top faces of the parts are Z0. This is the most commonly used method when using rotary devices. If more than two parts are located on the same side of the fixture they are each given a separate work offset.

4-Axis Machining Example

Below shows a simple block that needs to machined on 4 surfaces. The block has four ¼ in holes with 3/8 counter bore. Both 4" ends need to be machined to size and 5/16 holes drilled thru halfway each side. The back side needs holes drilled and tapped into the cross 5/16 holes.

3x4 Extension Block 1018 Steel
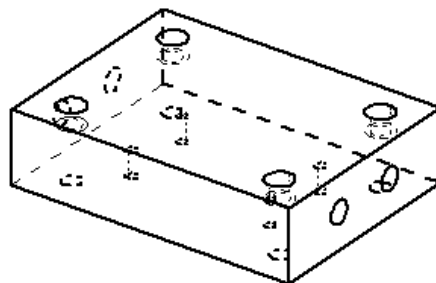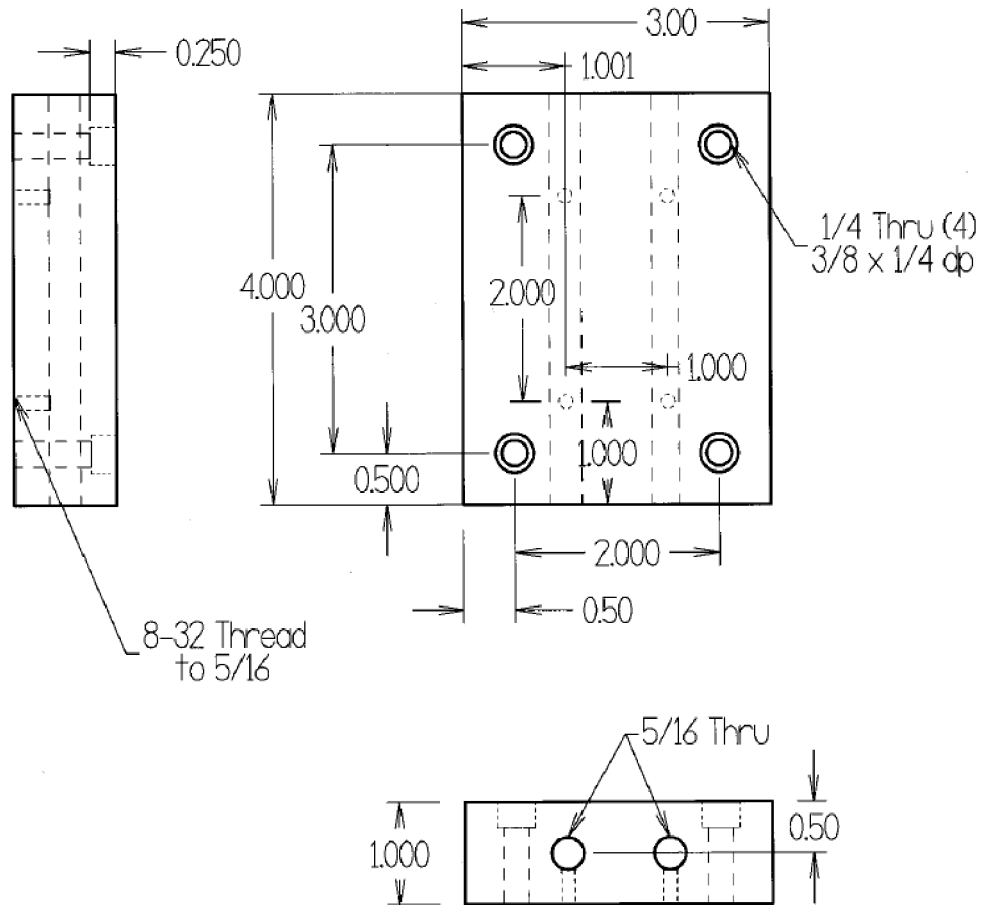
Below is a fixture designed by Productivity's Automation Department to hold two hydraulic vises. The fixture is attaches to a Haas HRT210 on the right side with an A frame support on the left side.
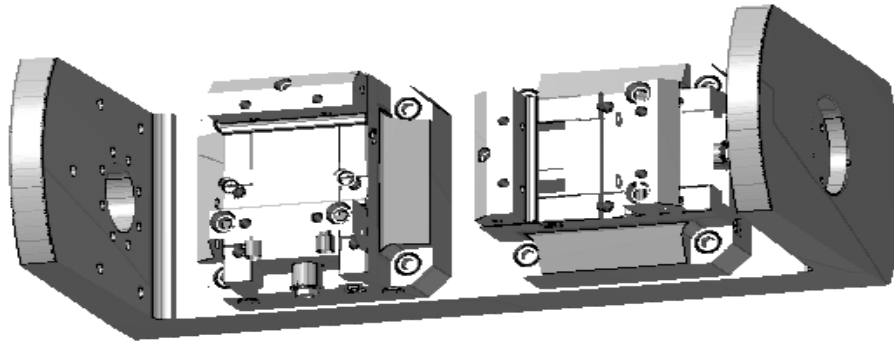

Below shows the block to be machined in the vises. The right vise has the top orthogonal view. The left vise has the back view but is rotated 90⁰.
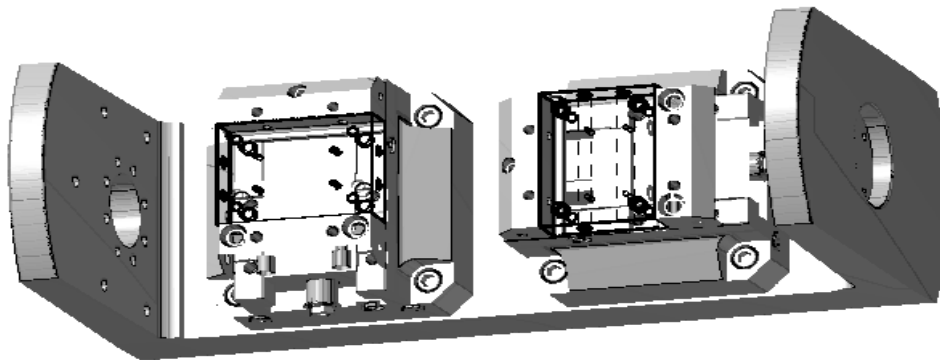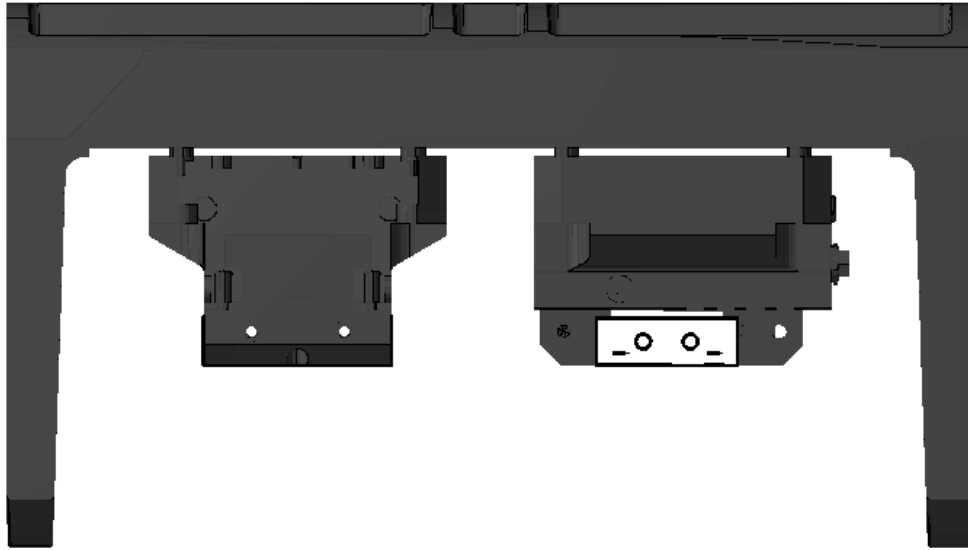

The ¼ thru hole with 3/8 counter bore may be machined in the right vise. The 8-32 thread drilled and tapped in vise on the left.

The rotary table is set up such that the front and back face may be machined as below. This angular position will be set up as **A0⁰**.
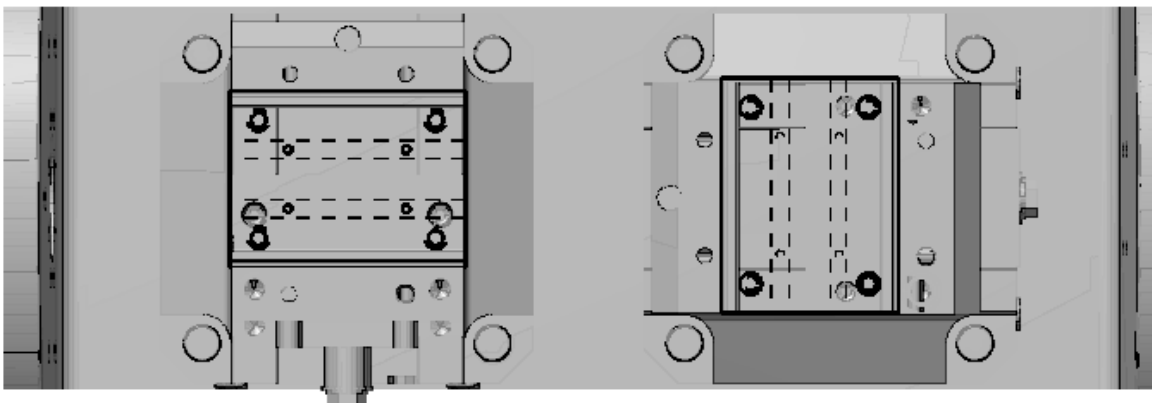
Below the **A = 90⁰.** In this position the end of the part may be faced off and the 5/16 hole drilled halfway thru the part.

Below the fixture is in the **A270⁰** position. The other end may be faced off to the overall length of 4 inches and the 5/16 hole drilled halfway to meet the hole drilled from position A90⁰.
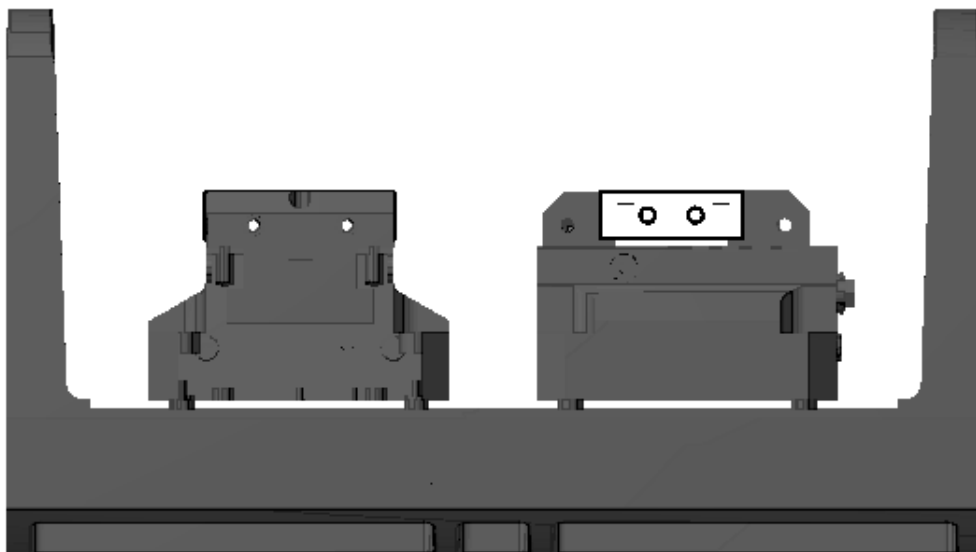
Machining Sequence:

Op 1: Face off end of part in right hand vise in fixture A90° position. Drill and chamfer 5/16 hole halfway thru part.

Op 2: Face off end of part in right hand vise in fixture position A270° position to overall length of 4". Finish chamfer and drill 5/16 thru hole to meet hole from Op 1.

Op 3: Drill ¼ hole thru part, counter bore 3/8 hole x ¼" deep.

Op 4: Drill and tap 8-32 thread thru to meet 5/16 hole. Chamfer ¼" holes.

Selection of Work Offsets: Multiple work offsets will be used to machine the block. Care should be taken in selection of work offsets. The part zero should be a position where all the dimensions come off of on the print.   In this example all the locations come off the lower left corner in the top orthogonal view of the print. G56 represents that position in the A0° position in the right vise. The part is flipped over with center of rotation the x axis. It is then rotated 90° counter clockwise. G57 represents the same part zero as G56 except on the back side. G55 in the A270° view is the same point as G56 in the A° position.   G54 (A90°) is the same corner except on the opposite end of the part as G55.

A90°

A270°

A0°
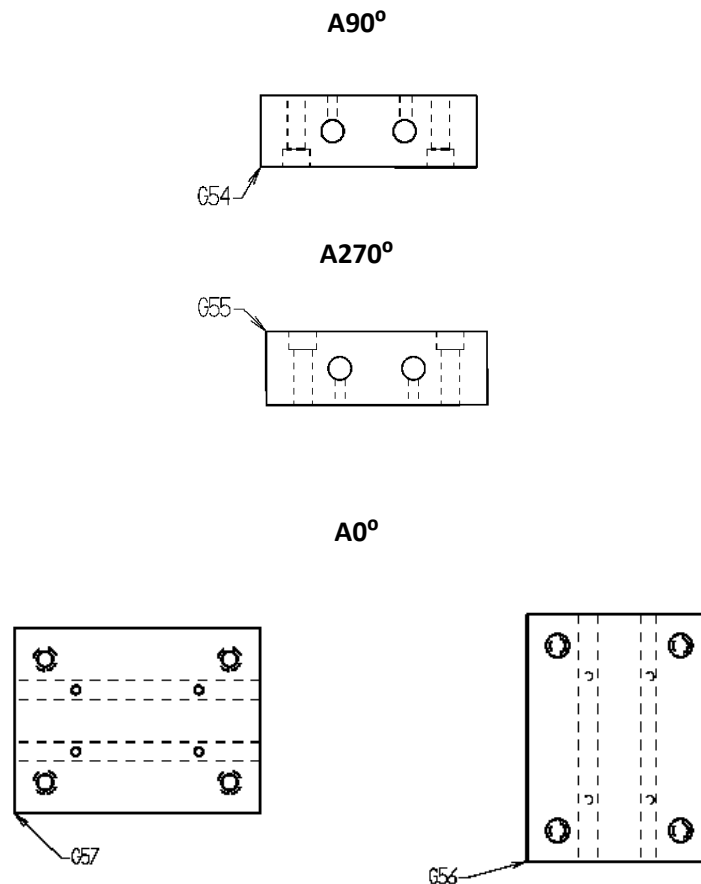
Program and tool selection:

Op 1 and Op 2: A 1 ½" Weldon Shank Kennametal index able end mill with three inserts is selected to face off the ends of the block.   According to Kennametal the ideal cutter diameter to part width or cut ratio is 1:1.5. For better tool life ¼ of the cutter diameter should be outside the work piece. This results in a negative angle of entry. (Kennametal Milling Catalog 6050 Inch, Copyright 2008, Kennametal inc., Latrobe, PA 15650, p.488)   Standard carbide 1/2" 90° spot drill and   5/16th jobber length 118° point HSS drill are used to create the 5/16 thru holes.

Op 3:   A ¼" carbide 135° split point screw machine drill and a carbide 4 flute, stub length 30° helix are used to create the counter bored holes .

Op 4 #29 cobalt screw machine length drill and 8-32 standard tap are used for the back side.

O00005 (Extension Block)
G00 G17 G40 G49 G80 G90 G98
G91 G28 A0
G91 G28 Z0
T1 M06 (1 1/2 IND END MILL)
G90 A90. (INDEX TO A270)
M10
G00 G90 G54 X3.85 Y0.375 M03 S1220 (480 FT/MN)
G43 H01 Z1. M08
Z0.1
G01 Z0 F50.
G01 X-0.85 F9.7 (.002/REV)
G00 Z1. M09
G91 G28 Z0 M05

T2 M06 (90 DEG DRILL)
G00 G90 G54 X1. Y0.5 M03 S1020
G43 H02 Z1. M08
G99 G82 Z-0.171 R0.1 F2. P0.2
X2.
G00 G80 Z1. M09
G91 G28 Z0 M05

T3 M06 (5/16 DRILL)
G00 G90 G54 X1. Y0.5 M03 S980
G43 H02 Z1. M08
G99 G83 Z-2.14 R0.1 F3.9 Q0.31
X2.
G00 G80 Z1. M09
G91 G28 Z0 M05
M11
G91 A-180. (INDEX TO A270 DEG)
M10
T1 M06 (1 1/2 IND END MILL)

M10
G00 G90 G55 X3.85 Y-1.375 M03 S1220 (480 FT/MN)
G43 H01 Z1. M08
Z0.1
G01 Z0 F50.
G01 X-0.85 F9.7 (.002/REV)
G00 Z1. M09
G91 G28 Z0 M05

T2 M06 (90 DEG DRILL)
G00 G90 G55 X1. Y-0.5 M03 S1020
G43 H02 Z1. M08
G99 G82 Z-0.202 R0.1 F2. P0.2
X2.
G00 G80 Z1. M09
G91 G28 Z0 M05

T3 M06 (1/4 DRILL)
G00 G90 G55 X1. Y-0.5 M03 S980
G43 H02 Z1. M08
G99 G83 Z-1.09 R0.1 F3.9 Q0.25
X2.
G00 G80 Z1. M09
G91 G28 Z0 M05
M11
G90 A0 (INDEX TO A0)
M10

T2 M06 (90 DEG DRILL)
G00 G90 G56 X0.5 Y0.5 M03 S1020 (RIGHT SIDE VISE)
G43 H02 Z1. M08
G99 G82 Z-0.202 R0.1 F2. P0.2
X2.5
Y3.5
X0.5
G00 G80 Z1. M09
G91 G28 Z0 M05

T4 M06 (1/4 DRILL)
G00 G90 G56 X0.5 Y0.5 M03 S1528
G43 H04 Z1. M08
G99 G83 Z-1.1 R0.1 F3. Q0.25
X2.5
Y3.5
X0.5
G00 G80 Z1. M09
G91 G28 Z0 M05

T4 M06 (3/8 END MILL)
G00 G90 G56 X0.5 Y0.5 M03 S1528
G43 H05 Z1. M08
G99 G82 Z-0.25 R0.1 F12.2 P0.25
X2.5
Y3.5
X0.5
G00 G80 Z1. M09
G91 G28 Z0 M05
(LEFT SIDE VISE G57)
T2 M06 (90 DEG DRILL)
G00 G90 G57 X1. Y1. M03 S1020
G43 H02 Z1. M08
G99 G82 Z-0.078 R0.1 F2. P0.2
Y2.
X3.
Y1.
X0.5 Y0.5 Z-0.14
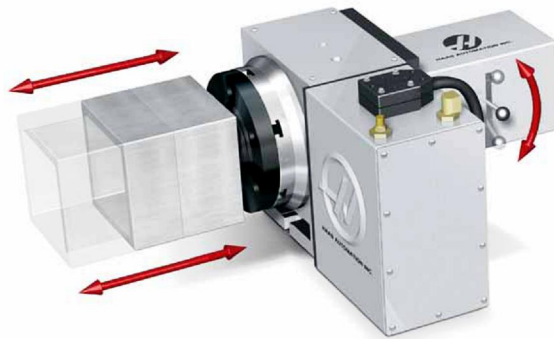Y2.5
X3.5
Y0.5
G00 G80 Z1. M09
G91 G28 Z0 M05

T5 M06 (8-32 TAP)
G00 G90 G57 X1. Y1. S790
G43 H02 Z1. M08
G99 G84 Z-0.62 R0.2 F24.87
Y2.
X3.
Y1.
G00 G80 Z1. M09
G91 G28 Z0 M05
G53 Y0
G53 X-1.
G28 G90 A0
M11
M30
%

Haas Quikchange Tooling Systems

QuikCube System

QuikPlate System

Tooling Block System

Thread Mill Exercise Solution

Thread Mill a 1.5 Diameter x 10 TPI through hole .5 deep. Use a .5 diameter 4 flute thread mill using cutter comp. SFM = 400   Feed = .004/rev. Hole is cut to 1.379, the thread minor.

$$RPM = \frac{3.82 \times SFM}{Diameter} = \frac{3.82 \times 400}{.5} = \mathbf{3056}$$

Inch/min = RPM x Feed x # flutes = 3056 x .004 x 4 = **48.9 in/min**

Internal Threads: $F1 = \frac{F2 \times (Dw - Dc)}{Dw}$ External Threads $F1 = \frac{F2 \times (Dw + Dc)}{Dw}$
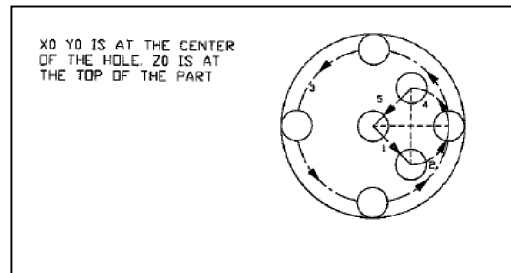
F1 = Programmed feed rate at center of tool
F2 = Actual feed rate desired at the cutting edge
Dw = Thread diameter
Dc = Cutter diameter

$$F1 = \frac{48.9 \times (1.5 - .5)}{1.5} = \mathbf{32.6\ in/min}$$

N1 T2 M6 (MILL THREAD)
N2 G00 G54 G90 X0 Y0 M03 S509   (Position to hole center in X and Y and turn on spindle)
N3 G43 H2 Z.1 M08 (Turn on tool length compensation, turn on coolant)
N4 G01 Z-.5125 F50. (Start depth plus 1/8 the pitch,
N5 G41 X.375 Y-.375 D2 (Activate cutter comp 45 degrees, ½ radius of finish diameter)
N6 G03 X.75 Y0 I0 J.375 Z-.5 F32.6 (Arc on to finish diameter radius ½ the radius of finish diameter)
N7 G03 I-.75 Z-.40 (Circular motion counter clockwise 360 degrees cutting thread, move Z positive one
             pitch length)
N8 G03 X.375 Y.375 I-.375 J0 Z.0125 (Arc off thread diameter ½ radius of the finish diameter, move up Z
                  1/ 8 x pitch length)
N9 G01 G40 X0 Y0 F50. (Cancel cutter comp, move to center of thread)
N10 G53 Z0 M9
N11 M30

References

Haas Automation Rotary/Tailstock Operator's Manual, December 2012, 96-0315 rev R, Haas Automation Inc., 2800 Sturgis Road, Oxnard, CA 93030, Tel. 888-817-4227 Fax 805-278-8561, www.HaasCNC.com

Haas Automation HA5C Operator's Manual, January, 2006, 96-4039 rev M, Haas Automation Inc., 2800 Sturgis Road, Oxnard, CA 93030, Tel. 888-817-4227 Fax 805-278-8561, www.HaasCNC.com

Haas Automation HRT Operator's Manual, January 2006, 96-5047 rev M, Haas Automation Inc., 2800 Sturgis Road, Oxnard, CA 93030, Tel. 888-817-4227 Fax 805-278-8561, www.HaasCNC.com

Haas Automation Rotary Brochure Spread 2011, Haas Automation Inc., 2800 Sturgis Road, Oxnard, CA 93030, Tel. 888-817-4227 Fax 805-278-8561, www.HaasCNC.com

Fitzpatrick, Michael *Machining and CNC Technology,* Update Ed. McGraw Hill 2011

Lynch, Mike *Advanced Computer Numerical Control Techniques for Machining & Turning Centers,* CNC Concepts, Inc 1992

Smid, Peter *CNC Programming Handbook,* 3rd Ed. New York, NY: Industrial Press 2008